Research Article

CFD Analysis Of Helical Screw Tape Of Heat Exchanger For Laminar Flow

[1]Ravi Shankar Kumar, [2]D.S. Chauhan

[1]M. tech scholar
R.D. Engineering College, Ghaziabad
[2]Assistant professor
R.D. Engineering College, Ghaziabad

Article History:Received:11 January 2021; Accepted: 27 February 2021; Published online: 5 April 2021

Abstract:The heat emission from heat exchangers is of great importance for the efficient and economical operation of industrial machines. The main focus of the study is to increase the overall heat transfer rate and optimize the design. And the Nusselt number and Reynolds number calculated using ANSYS Fluent. And the helical screw inserts save energy. A CFD study of the helical band insert with different twist ratio in the tube was performed for laminar flow as a heat transfer amplifier using commercial software. This use increased the rate of heat transfer with a significant increase in pressure drop. In this study, 3 types of helical belts were used to add value to HTC. The value of the friction factor increases with increasing Reynolds number. The value of HTC increases as the Reynolds number increases. The HTC goes up to 1562.5,1666,1886.79 in all three cases. The Nusselt number and Reynold number increase to 94.33 and 1997.5, respectively. In case 3 The temperature of the pipe wall is reduced if a pipe with an insert is used, which means less irreversibility and greater heat transfer properties. It has a relatively higher and lower temperature spiral profile in the pipe wall as the insert twist.

Keywords: CFD, Heat exchanger, helical screw, ANSYS.

I. Introduction

The heat output of heat exchangers is vital for the proficient and practical activity of modern machines. There are numerous dynamic and detached methods, for example, altering the fluid channel, adding vortex generators, mixing nanofluids to improve the warm exhibition of cylinder side convection heat move of heat exchangers. These methods lessen the irreversibility of the establishment, increment the quality and amount of heat move, and improve the stream properties of liquids. Specialists from around the planet have considered the improvement of warm execution of convective heat move in pipes utilizing different kinds of additions as liquid stream modifiers. A large portion of the addition calculation capacities as a vortex generator, blending and disturbance inductor and home time enhancer, typical speed inclinations. In the heat exchanger, the heat energy starting with one liquid is moved then onto the next liquid. Heat move can happen across the surface to the liquid or between strong particles to the liquid. The fundamental model for heat move between fluids or from the surface to the fluid is the warm inclination or the temperature contrast between the two. Heat exchangers are generally utilized for heating and cooling fluids. The cycle includes dissipation, buildup, and so forth Some different employments of heat exchangers incorporate sanitization, purification. The fluids in the heat exchanger are isolated by a divider with the goal that the fluids don't blend. These heat exchangers are called recuperator.

II. Literature review

DheerajShriwas et al. [1] in this paper presented are the Heat exchangers are used in various industries to improve heat transfer. This article looks at several techniques for improving heat transfer to discover new techniques for improving heat transfer rate.

Indri Yaningsih et al. [2] This item enhances the Nusselt number (nu) and friction factor (f) check as a heat check with different design ratios (w / w) of V-cut (VTT) twisted ribbon inserts. Three different width ratios (weight / weight) 0.32; In this experiment, 0.38 and 0.48 were introduced. The working fluid is hot water in the inner tube and cold water in the ring, the direction of flow is counter-current. Incoming hot water temperature is kept constant at 60 C, Reynolds flow rate # 5400 - 17,350. For comparison, the lower heat exchanger (soft tube) insert and the specific twisted ribbon insert (TT) are examined.

Tushar R. Shinde et al. [3] nowadays, heat transfer technology is used in many parts of the world in various heat exchange applications such as automobiles, refrigerators, air conditioners, process industries, etc. The fundamental goal of better heat transfer is to promote high heat fluxes. This mainly results in a reduction in the size of the heat exchanger, which results in a lower investment cost.

Suhas V. Patil et al. [4] The present work provides an summary of the research beat the past decade to enhance heat transfer during a circular tube and a square channel. within the present work, the main target is on performing on

twisted tape inserts and screw tapes, as these are known, consistent with recent studies, as a cheap tool within the field of warmth transfer performance improvement.
Nimish Dubey et al. [5] in the present work, optimal tube insert design changes were made to a previously designed tube insert, and its heat output was analysed using CFD software called ANSYS fluent.

III. Objective

- The main objective of the study increases the total heat transfer rate and optimized the design.
- Calculated the Nusselt number and Reynolds number by using of ANSYS Fluent.
- Helical screw tape inserts will cause of energy savings.
- Calculate the pipe wall temperature and heat transfer rate on pipe wall.
- Increase overall performance of the system will increases due to helical insert coils.

IV. Methodology

A. CFD Analysis Tube with Helical SCREW

This research presents the results of calculating the heat and flow phenomena occurring in pipes with bolted screws for a medium with supercritical parameters. The results were obtained by numerical analysis and CFD. CFD modeling was performed with the Ansys Fluent software package.

B. Preparation of model

Both the fluid domain geometry and the insert were designed in commercial software. The fluid domain has a length of 425 mm, a diameter of 30 mm. The cylindrical part of the insert is 19mm in diameter and the spiral insert wrapped around it is 4mm high and 1mm wide. The insert has a length of 350mm and a torque ratio of 1.92. It is 25 mm from the inlet rinse and ends 50 mm from the outlet. There is some distance between the insert and the pipe wall which means the insert is loose and convection heat transfer is the primary mechanism for transferring heat from the pipe wall to the water. If the insert is tight, there will be some conduction through the insert. To reduce complexity and computational effort, use is loosely appropriate. These two parts are then imported into ANSYS and the insert domain has been replaced by a Boolean cut in the adiabatic wall, which minimizes the computational domain. Here, the main purpose of the inserts is to act only as a vortex generator (i.e. a flow passage modifier). The domain is then discretized and solved with ICEM. The diagram of the computational model and the discretized domain is illustrated in Fig. 1.

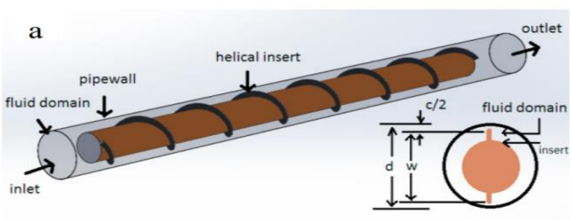


Fig. 1 Schematic of the 3D fluid domain with insert

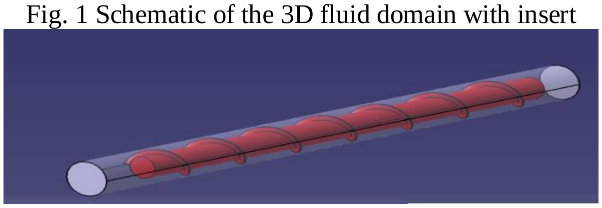


Fig. 2Catia model design of PIPE

C. Validations Methodology

1) First created cad model according base paper design data.

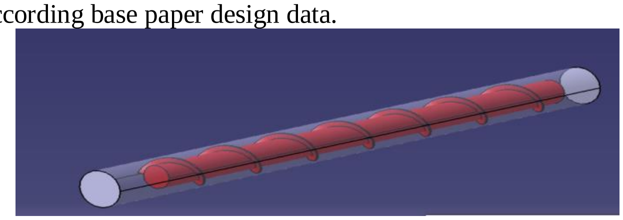


Fig. 3 Validations model (Design model for Validations)

2) Apply boundary condition according base paper for paper validations.

In a CFD study to improve heat transfer using an insert, heat input is implemented into the pipe wall using two methods. One is the constant flow of heat and another is the temperature. In this study, a heat input of 100,000 W / m2 is implemented in the pipe wall under uniform heat flow conditions. There are some assumptions such as steady state, fully developed incompressible laminar flow, no heat loss is used in this study. The pressure inlet and outlet velocity conditions are selected for the inlet and outlet respectively. The study is conducted for different water velocities for 0 <2000. Uniform velocity is used in the inlet which is fully developed in the pipe and zero pressure at the outlet to measure the pressure drop. The recessed walls are placed in adiabatic conditions assuming a negligible heat transfer. For the simple simulation of the pipe, only a fluid domain with a length of 425 mm and a diameter of 30 mm was used for the same heat flow and boundary conditions as a pipe with insert.

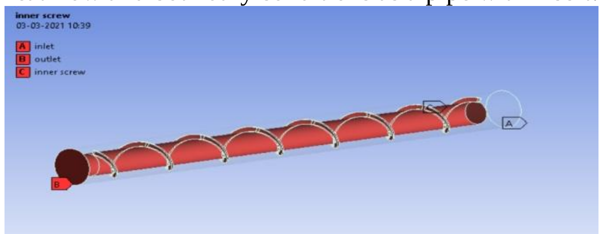


Fig.4.Boundary condition according base paper for validations

3) Results: -

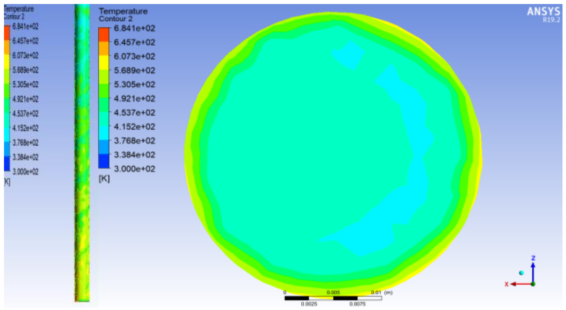


Fig. 5 Validations temperature contours

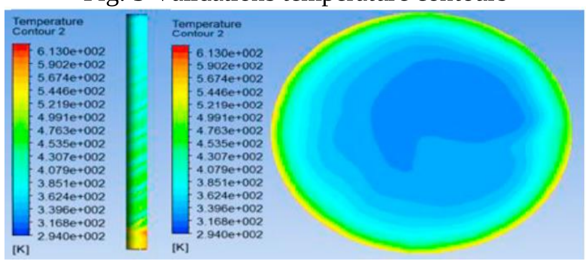


Fig. 6 Base paper result

	Validations Temperature	Base paper Temperature
Temperature	607k	613k

The paper outlet temperature 613k and validations temperature 607k.so total parentage of error during paper validations was 1%.

D. Steps of Working

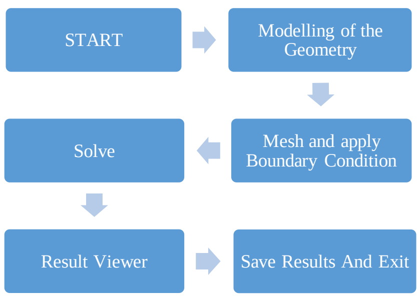


Fig. 7 Setup of working

E. Define step of working

1) First collection data from the base paper for paper validations.

2) The developing model by using cad software Catia. After developing mathematical model. The cad model imports into ANSYS workbench for CFD analysis.
3) Then apply meshing method and boundary conditions.
4) After meshing the fluent solver use to solve the CFD analysis.
5) Last final step calculated the results.
1. Import Model in ANSYS--
After developing cad model by using Catia f

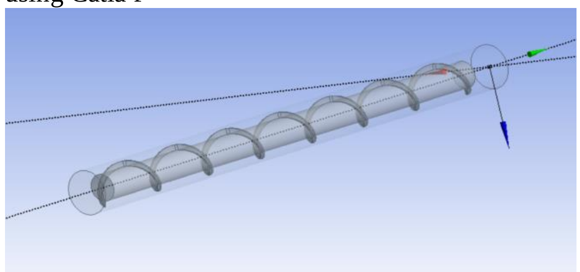


Fig. 8 Import domain Extract model in ANSYS

Case-1:-base model
Helical Pitch 50mm in case-1 and height and width was 4mm and 1mm.

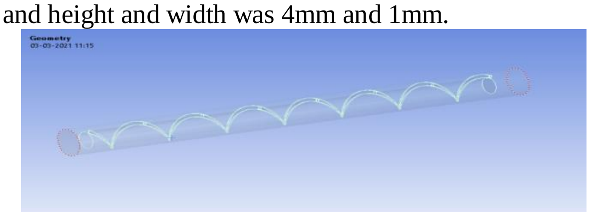


Fig. 9 Case-1 model

F. Meshing

In finite element analysis (FEA), the goal is to stimulate certain physical phenomena using a digital technique called the finite element method (FEM). To quantify physical phenomena such as wave propagation or the flow of liquids, we need to use mathematical equations. Most physical phenomena can be solved using partial differential equations (PDEs), but this is very difficult for most real-world problems. Any continuous object has infinite degrees of freedom (DOF) which makes it impossible to solve manual calculations. Then, in FEM, we create a mesh that divides the domain into a discrete number of elements for which the solution can be calculated. The data is then interpolated over the entire domain. A tetrahedral network was used in this study.

G. Why Tetrahedral mesh was used?

Computational Fluid Dynamics (CFD) simulation is becoming an important aspect in the construction and analysis of technical devices. With CFD, engineers can reduce losses associated with fluid flows, thereby increasing performance. The first part of the CFD simulation is to create a mesh adapted to the geometry to be analyzed. Tetrahedral mesh is often used when networking complex domains due to its flexibility. The tetrahedral mesh improves the accuracy of the results and offers the best result.

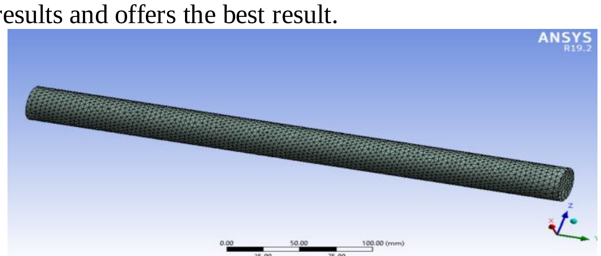


Fig.10 Meshing: Total No. of Nodes: 10049 & Total No. elements: 41032

Case-2:-helical pitch 40mm
Helical Pitch 40mm in case-2 and height and width was 4mm and 1mm

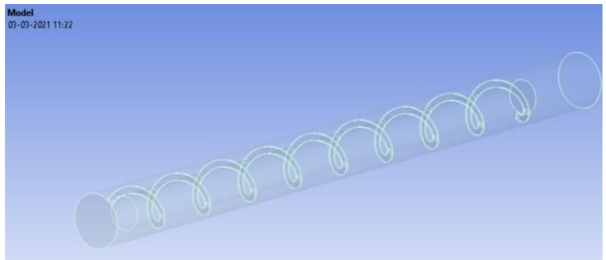


Fig. 11 Case-2 model

Case-2 MESHING

In finite element analysis (FEA), the goal is to simulate certain physical phenomena using a digital technique called the finite element method (FEM). To quantify physical phenomena such as wave propagation or the flow of liquids, we need to use mathematical equations. Most physical phenomena can be solved using partial differential equations (PDEs), but this is very difficult for most real-world problems. Any continuous object has infinite degrees of freedom (DOF) which makes it impossible to solve manual calculations. Then, in FEM, we create a mesh that divides the domain into a discrete number of elements for which the solution can be calculated. The data is then interpolated over the entire domain. A tetrahedral network was used in this study.

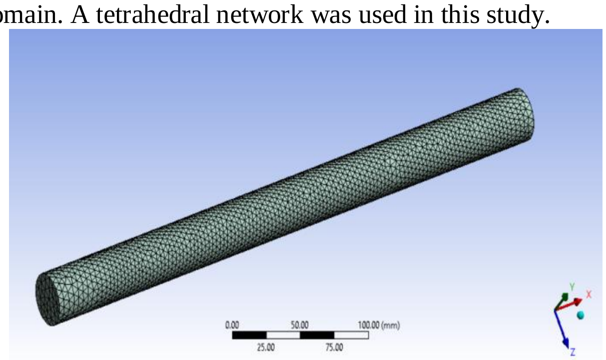


Fig. 12 Meshing: Total No. of Nodes: 10640 & Total No. elements: 43598

Case-3:-helical pitch 30mm

Helical Pitch 30mm in case-1 and height and width was 4mm and 1mm

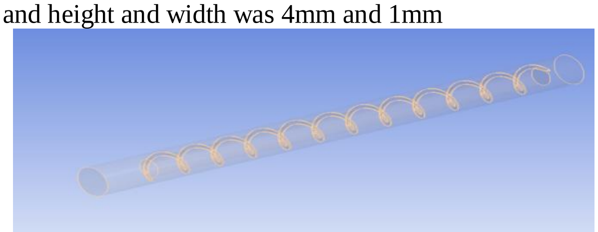


Fig. 13 Case-3 model

Case-3 Meshing

In finite element analysis (FEA), the goal is to simulate certain physical phenomena using a digital technique called the finite element method (FEM). To quantify physical phenomena such as wave propagation or the flow of liquids, we need to use mathematical equations. Most physical phenomena can be solved using partial differential equations (PDEs), but this is very difficult for most real-world problems. Any continuous object has infinite degrees of freedom (DOF) which makes it impossible to solve manual calculations. Then, in FEM, we create a mesh that divides the domain into a discrete number of elements for which the solution can be calculated. The data is then interpolated over the entire domain. A tetrahedral network was used in this study.

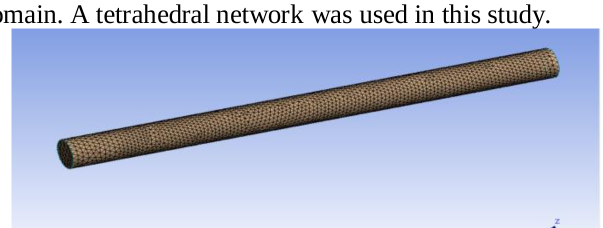


Fig. 14 Meshing: Total No. of Nodes: 10918 & Total No. elements: 44558

Table:-1 Design data

CASES	Helical height(mm)	Helical width(mm)	Pitch(mm)
CASE-1	4	1	50
CASE-2	4	1	40
CASE-3	4	1	30

Table 2: - Meshing Elements and Nodes for different Cases

CASES	Nodes	Elements
CASE-1	10049	41032
CASE-2	10640	43598
CASE-3	10918	44558

5. DEFINING MATERIAL PROPERTIES

- Water properties

Properties	Values
Density (kg/m^3)	998
Specific heat(j/kg-k)	4182
Thermal conductivity(w/m-k)	0.6

Table 3: - Water Properties

H. Boundary Condition

1. Define Name selection

In a CFD study to improve heat transfer using an insert, heat input is implemented into the pipe wall using two methods. One is the constant flow of heat and the other is the temperature. In this study, a heat input of 100,000 W / m2 is implemented in the pipe wall under uniform heat flow conditions. There are some assumptions such as a stable and fully developed incompressible laminar flow, no heat loss is used in this study. The pressure inlet and outlet velocity conditions are selected for the inlet and outlet respectively. The study is conducted for different water velocities for 0 <2000. In the inlet, which is fully developed in the pipe, a constant speed and no outlet pressure is used to measure the pressure drop. The recessed walls are placed in adiabatic conditions assuming a negligible heat transfer. For the simple simulation of the pipe, a single fluid domain with a length of 425 mm and a diameter of 30 mm was used for the same heat flow and boundary conditions as an insert pipe.

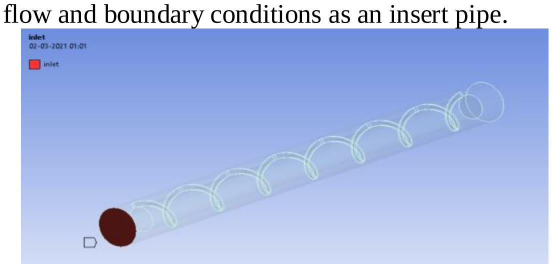


Fig. 15Define inlet conditions for CFD analysis

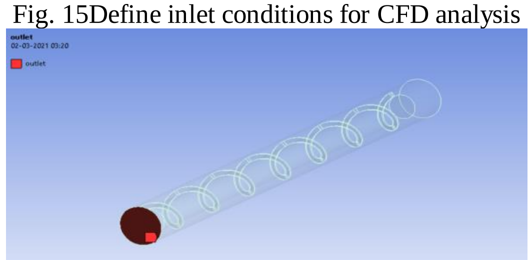


Fig. 16Define outlet conditions for CFD analysis

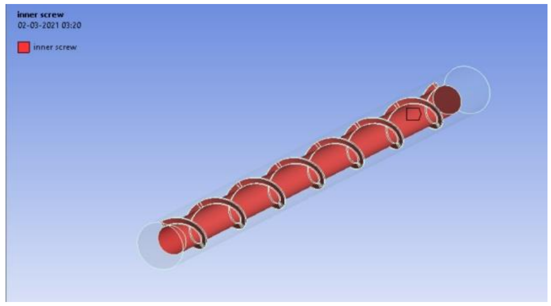


Fig. 17 Define boundary conditions

Boundary conditions	Values
Inlet Temperature	300k
Inlet velocity(m/s)	0.01,0.02,0.03,0.04,0.05
Outlet	Pressure outlet

I. Results

In this study, 3 types of helical screw inserts were used for the CFD analysis. In the first case the helical pitch value was 50 mm and in cases 2 and 3 the pitch value was 40 mm and 30 mm.

Case-1 helical pitch 50mm

Velocity contours:-The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 2.88e-02m/s. and minimum velocity is 8.65e-03m/s.

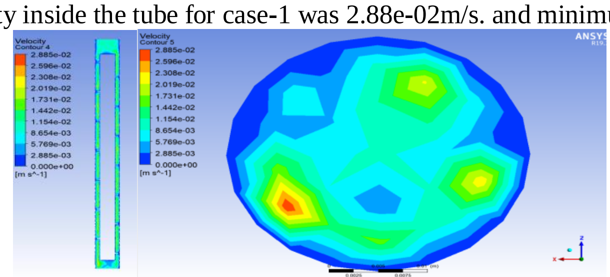


Fig. 18 Velocity contours

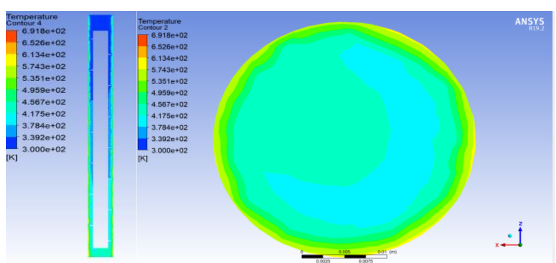


Fig. 19 Temperature contours top for case-1

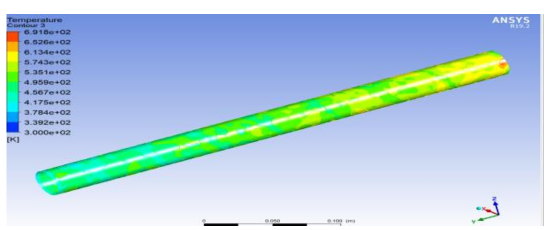


Fig. 20 Temperature contours for case-1

Temperature contours:-maximum temperature inside the tube was 691K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

Case-1 at inlet velocity 0.02m/s

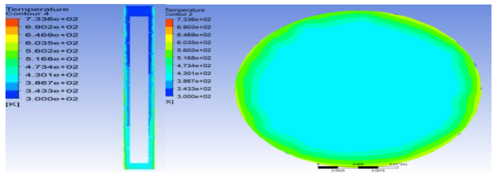


Fig. 21 Temperature contours at velocity 0.02m/s for case-1

The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 3.48e-02m/s. and minimum velocity is 3.4e-03m/s.

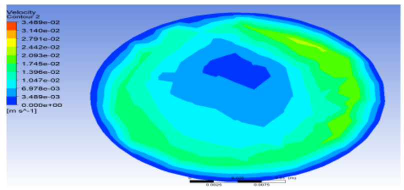


Fig. 22 Velocity contours at velocity 0.02m/s for case-1

Case-1 at inlet velocity 0.03m/s

The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 3.48e-02m/s. and minimum velocity is 3.4e-03m/s.

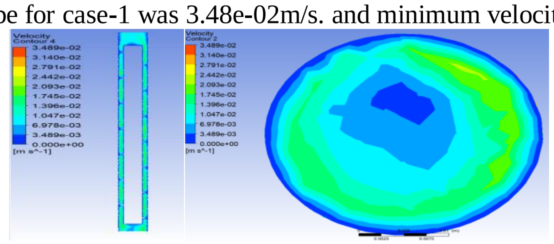


Fig. 23 Velocity contours at velocity 0.03m/s for case-1

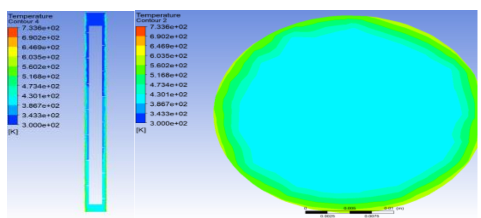


Fig. 24 Temperature contours at velocity 0.03m/s for case-1

Case-1 at inlet velocity 0.04m/s

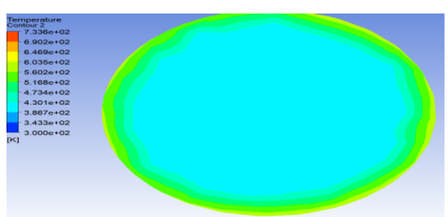


Fig. 25 Temperature contours at velocity 0.04m/s for case-1

The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 3.48e-02m/s. and minimum velocity is 3.4e-03m/s.

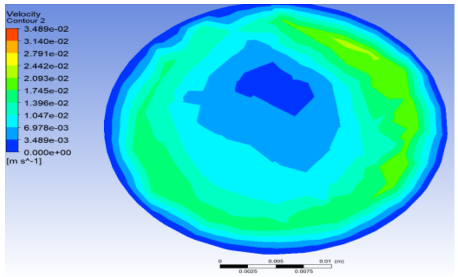


Fig. 26 Velocity contours at velocity 0.04m/s for case-1

Case-1 at inlet velocity 0.05m/s

Temperature contours:-maximum temperature inside the collector was 733K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

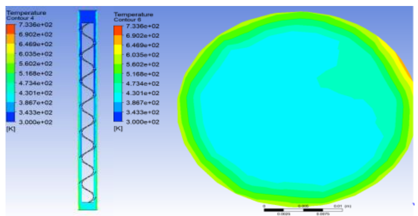


Fig. 27 Temperature contours at velocity 0.05m/s for case-1

The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 3.48e-02m/s. and minimum velocity is 3.4e-03m/s.

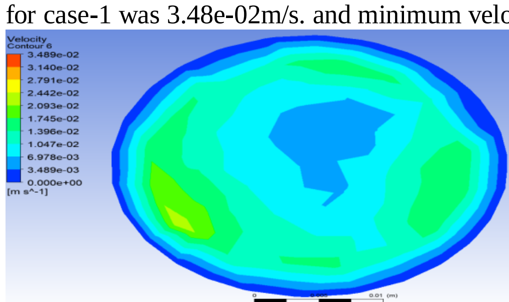


Fig. 28 Velocity contours at velocity 0.05m/s for case-1

Case-2 helical pitch 40mm

Temperature contours:-maximum temperature inside the collector was 657K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

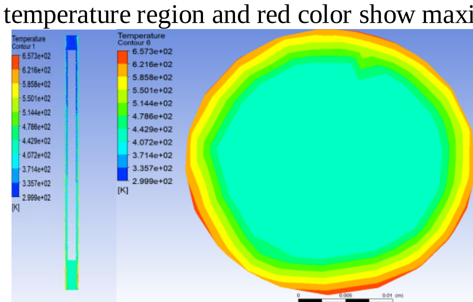


Fig. 29 Temperature contours for at velocity0.01m/s

Velocity contours:-The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 3.42e-02m/s. and minimum velocity is 3.4e-03m/s.

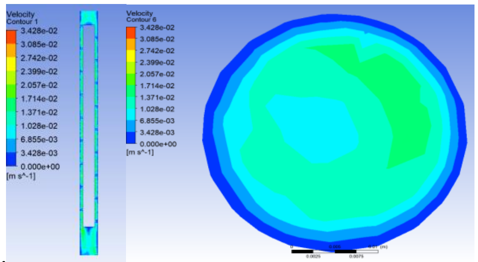


Fig. 30 Velocity contours at velocity 0.01m/s

Case-2 helical pitch 40mmat velocity 0.02m/s

Temperature contours:-maximum temperature inside the collector was 666K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

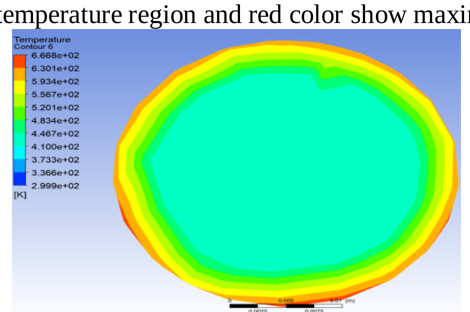


Fig. 31 Outlet Temperature contours for at velocity0.02m/s

The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 4.06e-02m/s. and minimum velocity is 4.e-03m/s.

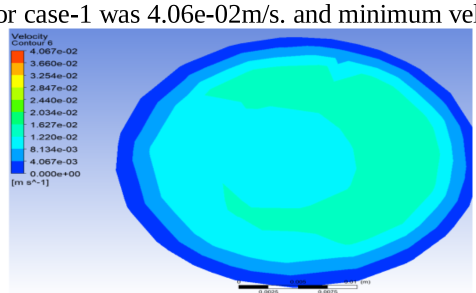


Fig. 32 Outlet velocity contours for at velocity0.02m/s

Case-2 helical pitch 40mmat velocity 0.03m/s

Temperature contours:-maximum temperature inside the collector was 666K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

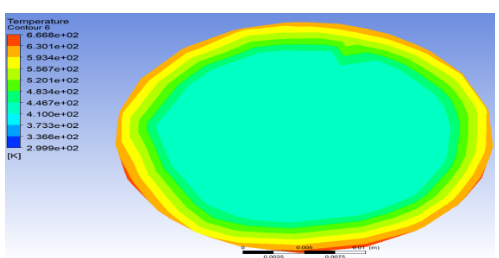


Fig. 33 Outlet Temperature contours for at velocity0.03m/s

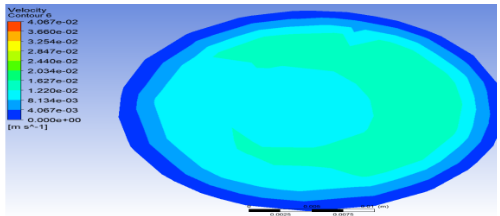


Fig. 34 Outlet velocity contours for at velocity0.03m/s

Case-2 helical pitch 40mmat velocity 0.04m/s

Temperature contours:-maximum temperature inside the collector was 666K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

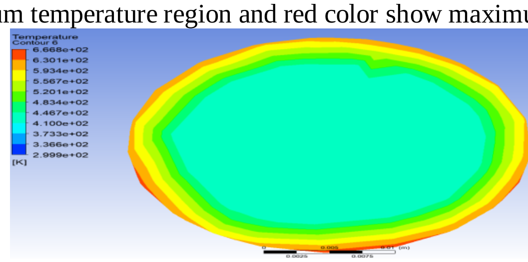


Fig. 35 Outlet Temperature contours for at velocity0.04m/s

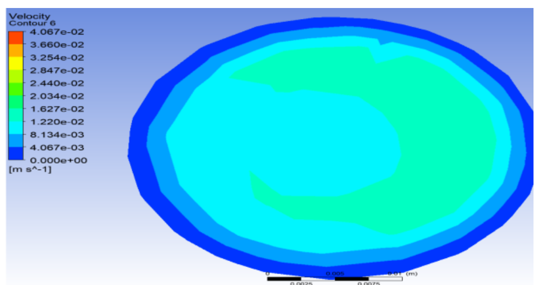


Fig. 36 Outlet velocity contours for at velocity0.04m/s

Case-2 helical pitch 40mmat velocity 0.05m/s

Temperature contours:-maximum temperature inside the collector was 666K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

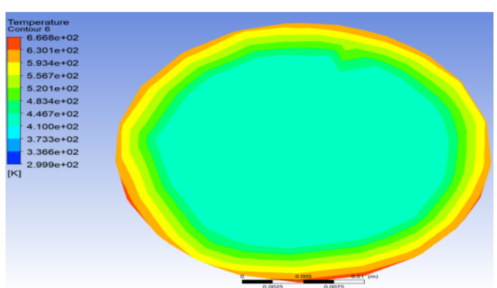


Fig. 37 Outlet Temperature contours for at velocity0.05m/s

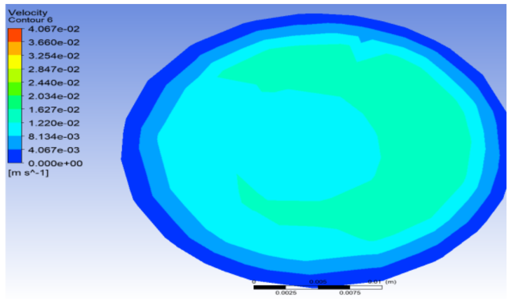


Fig. 38 Outlet velocity contours for at velocity0.05m/s

Case-3 helical pitch 30mm

Temperature contours:-maximum temperature inside the collector was 772K and minimum temperature 579k.the light blue color show the minimum temperature region and red color show maximum temperature region.

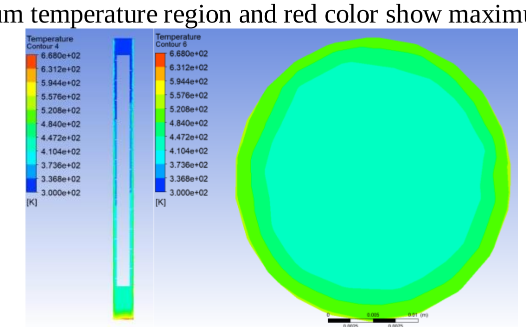


Fig. 39 Outlet Temperature contours for at velocity0.01m/s

Velocity contours:-The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 5.68e-02m/s. and minimum velocity is 5.68e-03m/s.

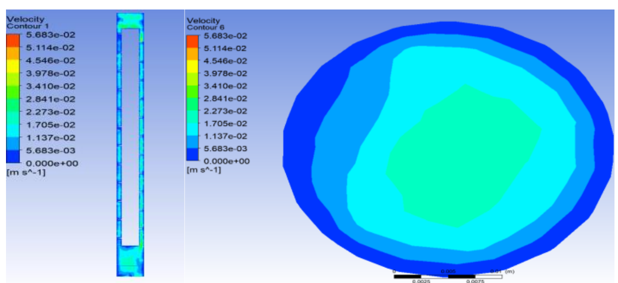


Fig. 40 Outlet velocity contours for at velocity0.01m/s

Case-3 Result at velocity at 0.02m/s

Temperature contours:-maximum temperature inside the collector was 504K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

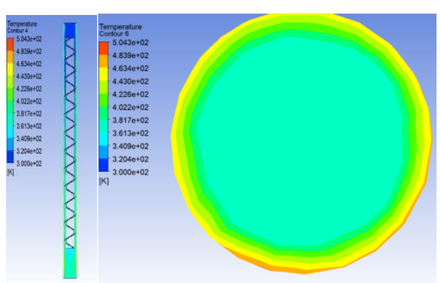


Fig. 41 Outlet Temperature contours for at velocity0.02m/s

The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 6.373e-02m/s. and minimum velocity is 6.373e-03m/s.

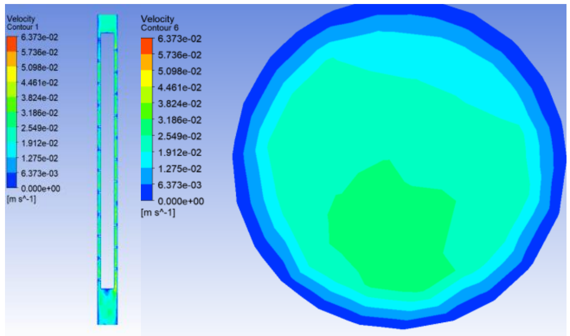


Fig. 42 Outlet velocity contours for at velocity0.02m/s

Case-3 Result at velocity at 0.03m/s

Temperature contours:-maximum temperature inside the collector was 478K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

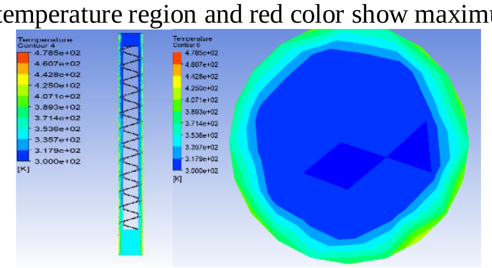


Fig. 43 Outlet Temperature contours for at velocity0.03m/s

The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 3.48e-02m/s. and minimum velocity is 3.4e-03m/s.

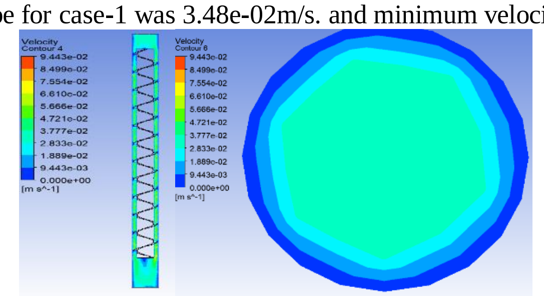


Fig. 44 Outlet velocity contours for at velocity0.03m/s

Case-3 Result at velocity at 0.04m/s

Temperature contours:-maximum temperature inside the collector was 480K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

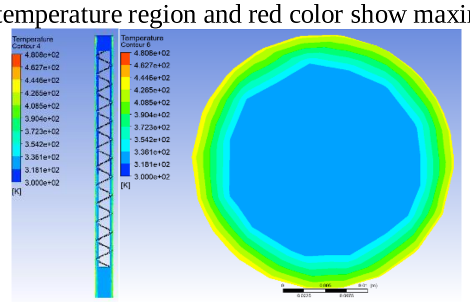


Fig. 45 Outlet Temperature contours for at velocity0.04m/s

The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 1.2e-1m/s. and minimum velocity is 1.2e-02m/s.

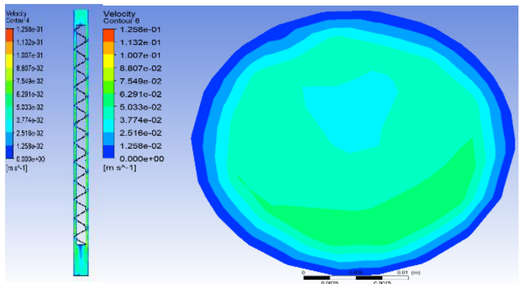


Fig. 46 Outlet velocity contours for at velocity0.04m/s

Case-3 Result at velocity at 0.05m/s

Temperature contours:-maximum temperature inside the collector was 483K and minimum temperature 300k.the light blue color show the minimum temperature region and red color show maximum temperature region.

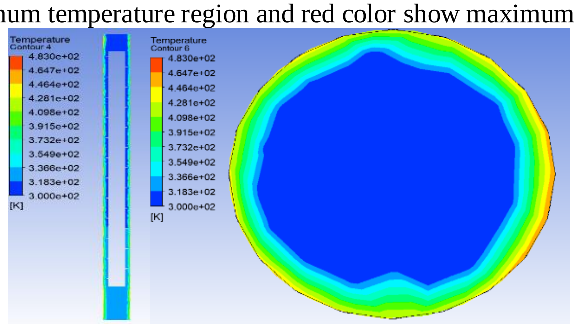


Fig. 47 Outlet Temperature contours for at velocity0.05m/s

The light blue color show the minimum velocity region and red color show maximum velocity region. The maximum velocity inside the tube for case-1 was 1.52e-1m/s. and minimum velocity is 1.5e-02m/s.

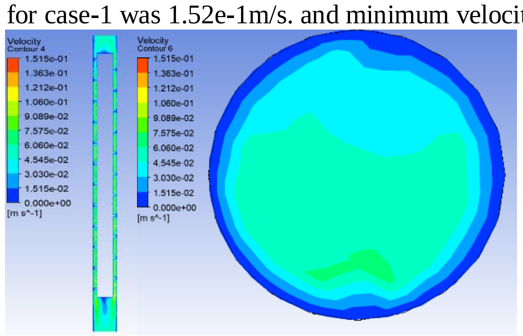


Fig. 48 Outlet velocity contours for at velocity0.05m/s

V. RESULT

A. Result Calculation

The effect of addition of insert and velocity for friction factor is plotted against Reynolds number in graph 5.10,5.11,5.12 It is visible that the simulated and theoretical model for friction factor has similarity in trend. Friction factor drops as velocity increases which is similar in nature reported by Sivashanmugam and Suresh.

Convective heat transfer coefficient is calculated as, where q is heat flux

$h = \frac{q}{T_s}$

Where q is 10000w/m^2(base paper)

D =30mm=.03

K=.6

Nu=hD/k = 425.53 * .03/ .6 = 9.13

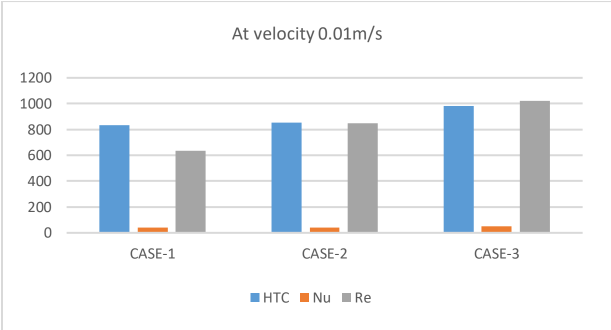
L=0.425m length of tube

Velocity in case first;-0.001m/s

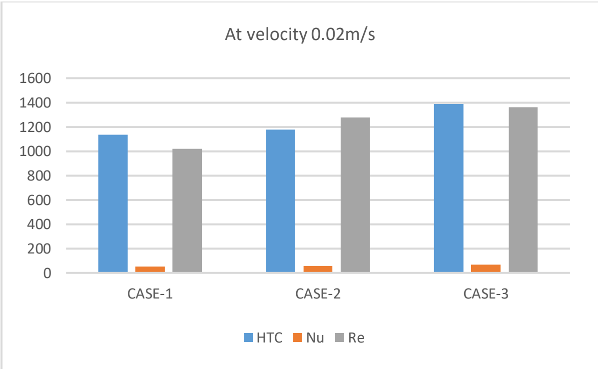
Re=0.425*0.001/(10^-6)=425

Friction factor is obtained

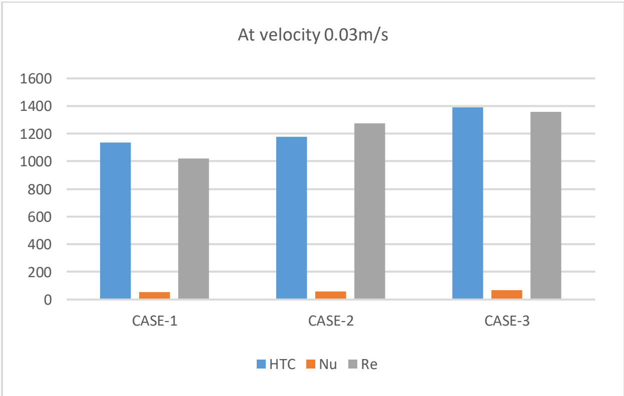
$$f = \frac{64}{Re}$$



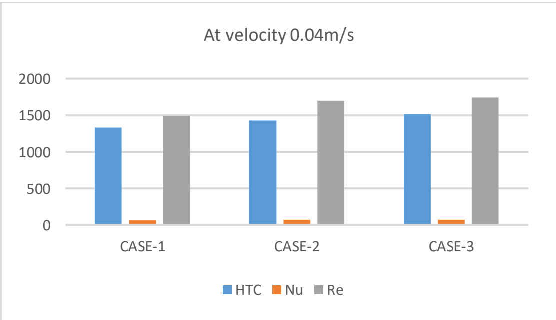
Graph 1: HTC and Nu, Re at velocity 0.01m/s



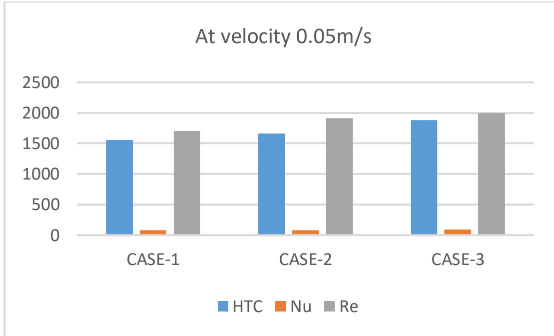
Graph 2: HTC and Nu, Re at velocity 0.02m/s



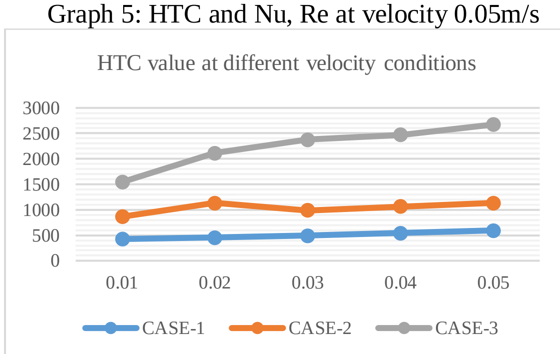
Graph 3 HTC and Nu, Re at velocity 0.03m/s



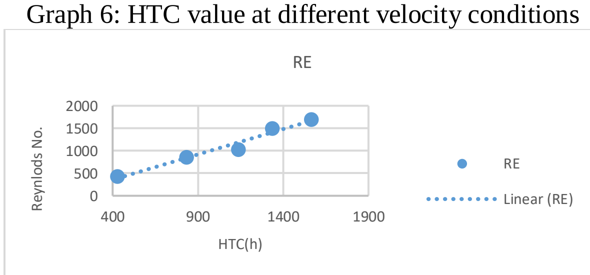
Graph 4: HTC and Nu, Re at velocity 0.04m/s



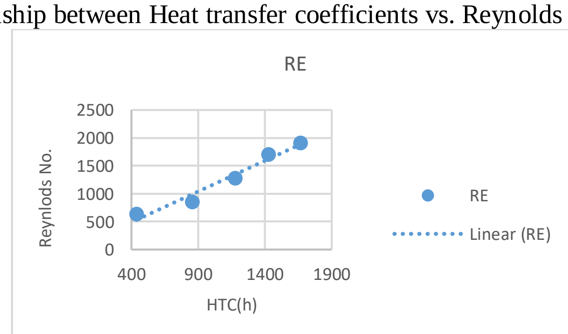
Graph 5: HTC and Nu, Re at velocity 0.05m/s



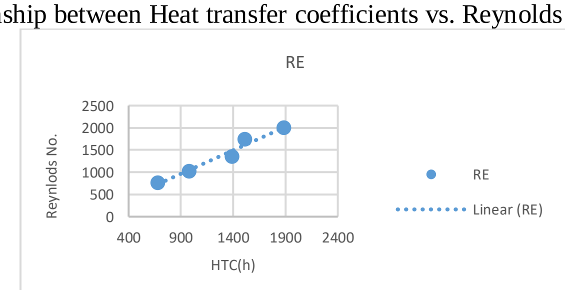
Graph 6: HTC value at different velocity conditions



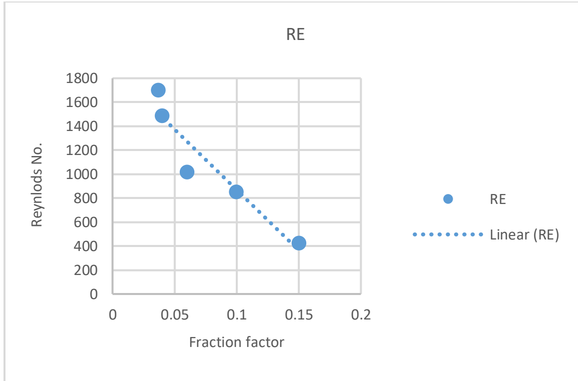
Graph 7 Relationship between Heat transfer coefficients vs. Reynolds number For Case-1



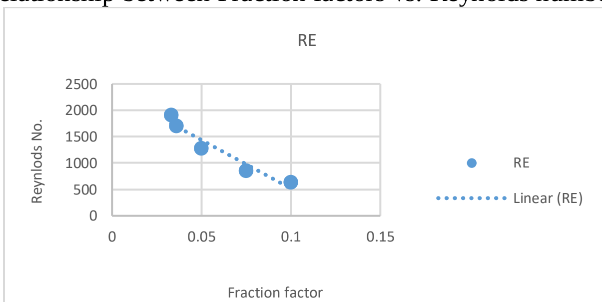
Graph 8 Relationship between Heat transfer coefficients vs. Reynolds number For Case-2



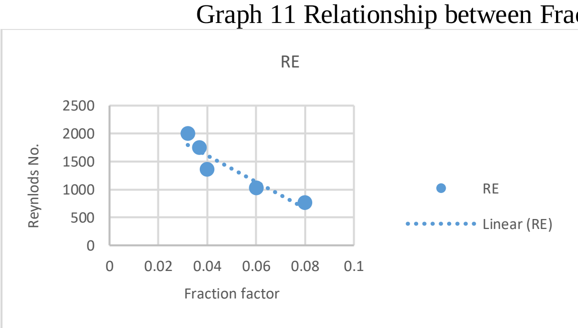
Graph 9 Relationship between Heat transfer coefficients vs. Reynolds number For Case-3



Graph 10 Relationship between Fraction factors vs. Reynolds number For Case-1



Graph 11 Relationship between Fraction factor vs. Reynolds number For Case-2



Graph 12 Relationship between Fraction factors vs. Reynolds number For Case-3

VI. CONCLUSION

A CFD study of helical screw tape insert of different twist ratio in tube was done for laminar flow as heat transfer enhancer using a commercial software package. This insert augmented the rate of heat transfer with significant increase in pressure drop. The findings of the study are:

- The Reynold number increases with increasing inlet velocity of helical screw tape.
- In this study 3 type of helical screw tape insert was used for increase the value of HTC.
- The value of Friction factor increase with increase the Reynold number.
- The value of HTC is increased with increase in Reynolds number. The HTC increases up to 1562.5,1666,1886.79 in all three cases.
- The Nusselt number and Reynold number increase up to 94.33 and 1997.5. in case-3
- Pipe wall temperature is reduced in case of insert fitted tube meaning lower irreversibility and the higher heat transfer characteristics.
- It has helical contour of relatively higher and lower temperature in pipe wall like the twist of the insert.

REFERENCES

1. DheerajShriwas, Mr. JagdeeshSain "Heat Transfer Enhancement Technique in Heat Exchanger: An Overview", 2018 IJRTI | Volume 3, Issue 9 | ISSN: 2456-3315.

2. Indri Yaningsih, Agung Tri Wijayanta "V-cut Twisted Tape Insert Effect on Heat Transfer Enhancement of Single Phase Turbulent Flow Heat Exchanger", AIP Conference Proceedings 1931, 030038 (2018); https://doi.org/10.1063/1.5024097.
3. Pittala, Suresh, and T. Diriba. "Computational Fluid Dynamics Analysis of Impeller Design For A Pump." International Journal of Mechanical Engineering (IJME) 5.4.
4. Tushar R. Shinde, Prof. D. G. Kumbhar "A Review on Experimental and CFD Analysis for Heat Transfer Enhancement in Heat Exchanger Tube using Drilled Twisted Tape Inserts of various Geometries", International Research Journal of Engineering and Technology (IRJET), Volume: 04 Issue: 06 | June -2017.
5. Reddy, P. RAVINDER, and M. Saikiran. "Aerodynamic Analysis of Return Channel Vanes in Centrifugal Compressors." International Journal of Mechanical Engineering (IJME), 5 (1), 73 82 (2016).
6. Suhas V. Patil and P. V.VijayBabu "Heat Transfer Augmentation in a Circular tube and Square duct Fitted with Swirl Flow Generators: A Review", International Journal of Chemical Engineering and Applications, Vol. 2 , No. 5 , October 2011.
7. Khan, Sher Afghan, Abdul Aabid, and Maughal Ahmed Ali Baig. "CFD analysis of CD nozzle and effect of nozzle pressure ratio on pressure and velocity for suddenly expanded flows." International Journal of Mechanical and Production Engineering Research and Development 8 (2018): 1147-1158.
8. Nimish Dubey, Morrish Kumar "CFD Analysis of Fluid Flowing Through a Heat Exchanger Tube Having a Twisted Tape with a Centrally Placed Semi-Circular Groove", International Journal of Science and Research (IJSR), Volume 6 Issue 6, June 2017.
9. Krishnaveni, J., G. Sowmya, and U. Sudhakar. "Thermal Analysis Of Cylinder Head By Using Finite Element Analysis." International Journal of Mechanical Engineering (IJME) ISSN (P), 2319 2240 (2014).
10. ShubhamJadhav, Jayesh Jain "Heat Transfer Augmentation Technique Using Twisted Tape Insert", International Conference on Ideas, Impact and Innovation in Mechanical Engineering (ICIIIME 2017), Volume: 5 Issue: 6 2017.
11. OrhanKeklikcioglu, and VeyselOzceyhan "A Review of Heat Transfer Enhancement Methods Using Coiled Wire and Twisted Tape Inserts", 2018.
12. Umesh, K. S., V. K. Pravin, and K. Rajagopal. "Experimental Investigation and CFD Analysis of Multi-Cylinder Four Stroke SI Engine Exhaust Manifold for Optimal Geometry to Reduce Back Pressure and to Improve Fuel Efficiency." International Journal of Automobile Engineering Research and Development, 4, 13 20 (2014).
13. SiddharthSoni, Prof. Brijendra Kumar Yadav "Heat Transfer Enhancement in Tubular Heat Exchanger Using Twisted Tape", International Journal of Science, Engineering and Technology, 2020.
14. Nikam, Harshal, P. R. E. M. Mishra, and SayaliBharambe. "Design and Analysis of Brake Rotor with Parameter Optimization." International Journal of Automobile Engineering Research and Development (IJAuERD) 4.4 (2014): 21-30.
15. SmithEiamsa-ard, PongjetPromvonge "Heat transfer characteristics in a tube fitted with helical screw-tape with/without core-rod inserts", International Communications in Heat and Mass Transfer, Volume 34, Issue 2, February 2007, Pages 176-185.
16. KwanchaiNanan, MonsakPimsarn "Heat transfer enhancement by helical screw tape coupled with rib turbulators", Journal of Mechanical Science and Technology volume 28, pages4771–4779(2014).