GRD Journal for Engineering | Volume 2 | Issue 6 | May 2017
ISSN: 2455-5703

Modelling and Analysis of Straight Tube and Helical Tube Heat Exchanger using Computational Fluid Dynamics

Santosh Km
Department of Mechanical Engineering
MACE, Kothamangalam

Dr. Benny Paul
Professor
Department of Mechanical Engineering
MACE, Kothamangalam

George Mathew
Professor
Department of Mechanical Engineering
MACE, Kothamangalam

Sabu Kurian
Professor
Department of Mechanical Engineering
MACE, Kothamangalam

Dr. Cibu K Varghese
Professor
Department of Mechanical Engineering
MACE, Kothamangalam

Abstract

Heat exchangers find extensive use in industrial applications such as automotive systems, power generation, nuclear industry, process plants, heat recovery systems, refrigeration systems, food industry, etc. A heat exchanger is a device where two fluids streams come into thermal contact in order to transfer the heat from hot fluid to cold fluid stream. It has been widely reported in literature that heat transfer rates in helical coils are higher as compared to those in straight tubes. Due to the compact structure and high heat transfer coefficient, helical coil are a preferred means of heat transfer which is finding application in the industry. In this investigation the aim is to identify the effects of variation of Helical Pitch, the helical diameter and the effect of orientation on the heat transfer rate and efficiency for helical heat exchangers

Keywords- CFD, Radiators, Helical Tube, Heat Exchanger, Straight Tube

I. INTRODUCTION

Most modern cars use aluminium radiators. These radiators are made by brazing thin aluminium fins to flattened aluminium tubes. The coolant flows from the inlet to the outlet through many tubes mounted in a parallel arrangement. The fins conduct the heat from the tubes and transfer it to the air flowing through the radiator.

The tubes sometimes have a type of fin inserted into them called a tabulator, which increases the turbulence of the fluid flowing through the tubes. If the fluid flowed very smoothly through the tubes, only the fluid actually touching the tubes would be cooled directly. The amount of heat transferred to the tubes from the fluid running through them depends on the difference in temperature between the tube and the fluid touching it. So if the fluid that is in contact with the tube cools down quickly, less heat will be transferred. By creating turbulence inside the tube, all of the fluid mixes together, keeping the temperature of the fluid touching the tubes up so that more heat can be extracted, and all of the fluid inside the tube is used effectively.

Datta N. Mehtreet.al [2] experimentally studied the thermal performance of car radiator usingAl_2O_3–nano fluid in temperature ranges from (40-75°C) under different fractions of nanoparticles from 0.5, 1, and 1.5% by volume. In their study, the heat transfer with water based nano-fluids was experimentally compared to that of pure water as coolant in an automobile radiator .Liquid flow rate in the range of 50lph to 200 lph and air velocity in the range of 3.8 m/s to6.2 m/s were chosen respectively. The fluid inlet temperature was varied from40°C to 75°C to find the optimum inlet condition. Results demonstrate that increasing coolant flow rate can improve the heat transfer performance. Also increasing the air flow rate improves the heat transfer rate. The rate of heat transfer enhancement was found19% to 42% in comparison with pure water.

D. Bohneet.al [3] measured thermal conductivity, density, and viscosity of ethylene glycol - water mixtures. The measurements have been performed in the temperature range from - 20°C to 1 8 0 T for thermal conductivity, from - 10°C to 150°C for density, and from - 10°C to 100°C for viscosity.

G.N. Tiwariet.al [4] the study of heat and mass transfer during natural convection heating for preparation of Jiggery was carried out for the open and closed conditions. An indoor experiment was conducted for simulation of developed thermal model for heated mass transfer for maximum evaporation. Evaporated water was condensed atthe inner surface for the closed system as fresh water. The effect of different rates of heating (varying voltage) and heat capacity of sugar cane juice on heat and mass transfer

All rights reserved by *www.grdjournals.com*

were also carried out. It was observed that the evaporative heat transfer coefficient depends significantly on the rate of heating and heat capacity.

R. Paul Linga Prakashet.al [5] did an experiment on Radiator plays a vital role in engine cooling system. When increasing the cooling efficiency of radiator causes increase the life time of engine. The efficiency of the radiator can be increased by changing the surface area or dimension of the tube or increasing the number of fins/tubes. The heat transfer rate for the existing radiator could be analysed. Findings: After analysing the existing radiator, the new radiator has been designed. Two flat plates are placing inside the tube which acts as the nozzle. Hence, nozzle velocity increases and pressure decreases. Pressure is directly proportional to temperature. Application/Improvements: Thus, the temperature of coolant inside the radiator decreases. As a result efficiency of the proposed radiator is increased 5.37% when comparing with existing method.

II. MATERIALS AND METHODS

A. Experimental Setup

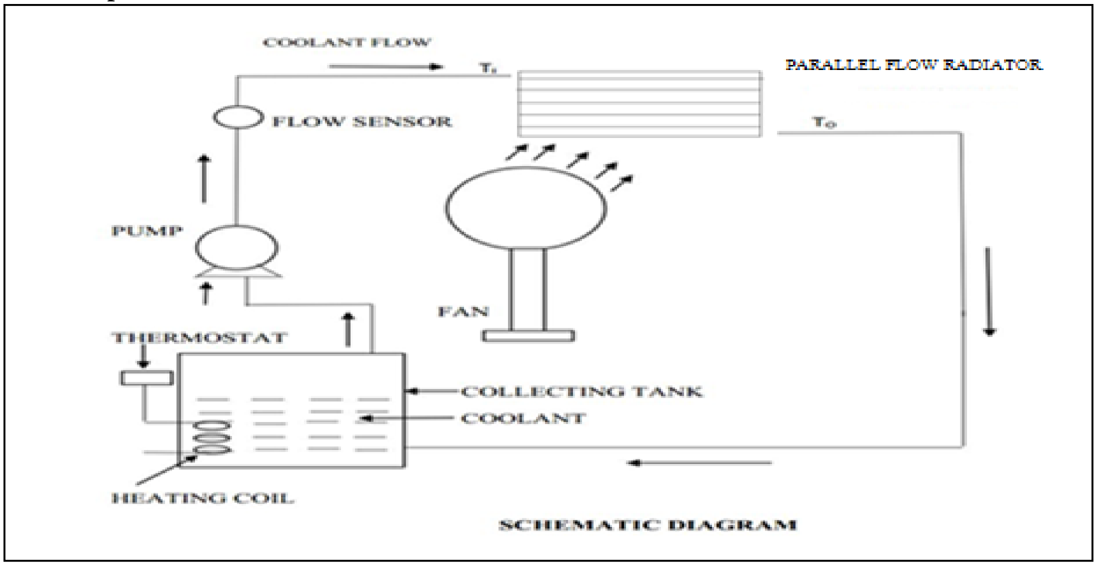

Fig. 1: Schematic Representation

Fig. 2: Experimental setup

This experimental setup includes a reservoir in the form of plastic tank, electrical heater, a centrifugal pump, a flow meter sensor, flow control valve, fan, thermostat ,infra-red thermometer for temperature measurement, wind gauge meter and heat exchanger (automobile radiator). The fluid flows through aluminium tube by a centrifugal pump from the tank to the radiator. The total volume of the circulating fluid is (5 litres) and constant in all the experimental steps. An electrical heater (1500W) is placed inside a plastic storage tank which represent the engine. The heater is used to heat the working fluid. A thermostat is provided to maintain

All rights reserved by *www.grdjournals.com*

inlet fluid temperature from 40 ºC to70ºC. A flow meter sensor and a ball valve is used to measure and control the flow rate of the fluid. An infra-red thermometer is used for recording the inlet and outlet fluid temperatures.

B. Radiator Specifications

Table 1: Radiator specifications

Material	*Aluminium*
Radiator Size	*340mm 140mm*
Radiator Fan	*Axial Fan*
Type	*Parallel*

C. Water

Total volume of coolant taken is 5 litres. Using of water alone as a coolant is obsolete. When water is used as coolant we have to regularly check the coolant level and refill it .The bad side of using water alone as coolant is it results in corrosion

D. Procedure

Consider experiment done using Water as coolant with boundary conditions as air inlet temperature 300K, coolant inlet temperatures taken are 323,326,321 and 337 K, coolant mass flow rate 0.0135 Kg/s and air inlet velocity 3.6 m/s. Ambient air temperature is taken as temperature of inlet air. Coolant inlet and outlet temperature is measured using an infrared thermometer whereas coolant mass flow rate is obtained using a flow sensor connected in coolant flowing line. Air velocity is measured using a wind gauge anemometer.

When the system is switched on, coolant starts flowing through the radiator. Heating coil starts heating the coolant in the collecting tank to a pre-set value in the thermostat. Radiator fan will start rotating. Flow sensor will record the coolant flow rate. Flow rate can be adjusted using a ball valve provided just below the flow sensor. Temperature at radiator inlet and outlet can be measured using infrared thermometer. And will have to wait for at least one hour for the temperature at radiator inlet and outlet to become steady.

E. CFD Setup

1) Model Description

For CFD analysis geometry was developed in Solid works 2016. Meshing as well as simulation is done in Ansys Workbench 16.2.Validation is done experimentally.

2) Geometric Details

- Number of Tubes: 3
- Pitch: 26.667mm (number of Turns : 12)
- Total Domain Extent for Helical Tube

1) Length : 320mm
2) Height : 140mm
3) Depth : 50mm

- Air Domain Extents

1) Length :680 mm
2) Height : 200mm
3) Depth : 420 mm

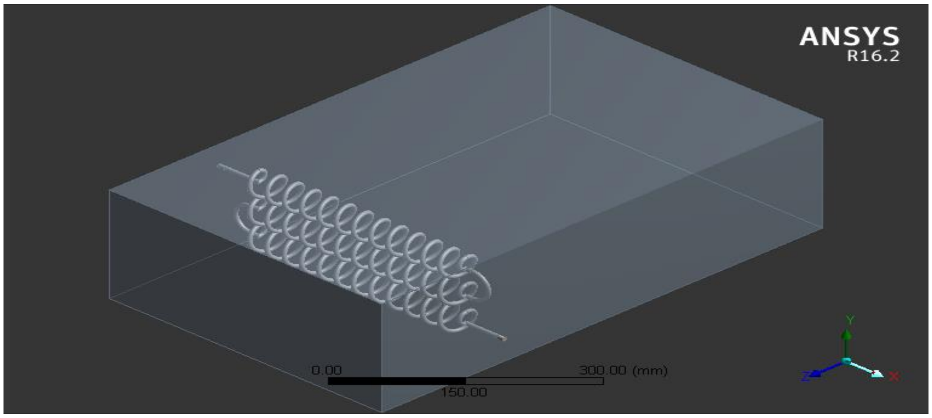

Fig. 3: Geometry

All rights reserved by www.grdjournals.com

This image is shows the geometry of the helical coil tube and the rectangular box like structure provide the geometry of the air domain.

1) Air Domain:
- Inlet : Velocity Inlet: Normal Velocity = 3.6m/s
- Outlet : Normal Pressure Outlet: 1 atm Operating
- Wall : Free Slip Wall or Zero Shear Condition
2) Water Domain
- Inlet : Mass Flow Inlet: 0.0135 kg/m^3
- Outlet : Normal Pressure Outlet: 1 atm Operating
- Wall : No Slip Wall Condition
3) Interface
- Physics Model : Conservative Heat Flux or Coupled Wall Condition
- Thin Material : Aluminum
- Thicknes : 0.001 m or 1 mm

F. Assumptions
- Velocity and temperature at the entrance of the radiator core for air and coolant is uniform.
- No phase change occurs in fluid streams.
- Fluid flow rate is uniformly distributed through the core in each pass on each fluid side.
- No flow leakages occur in any stream.
- The thermal conductivity of the solid material is constant.
- No internal source exists for thermal-energy generation

G. Validation

Table 2: Validation values

	Experimental Results	*Computational Results*
Inlet Water Temperature	*323 K= 50 ^{0}C*	*323 K = 50 ^{0}C*
Outlet Water Temperature	*316 K = 43 ^{0}C*	*315.076 K =42.076 ^{0}C*
Error Percentage	*2.15% Variation From Experimental Results*	

The CFD results are validated using experiment. It is found that only a variation of 2.15% is found for outlet water temperature when CFD and experimental results are compared.In both case we given a inlet tube temperature is 323 k and we obtain in the case of eceperimental output tube temperature is 3.16 k and in the case of CFD weget the value of 315.075.so that the error percentage is 2.15%.

III. RESULTS AND DISCUSSIONS

A. Experimental Results
From CFD results outlet coolant temperature is obtained. Using them the following tables are formulated. And heat transfer is calculated using

Table 3: Experimental Readings

Sl no	*Thermostat temperature (K)*	*Tube temperature (K)*		*Coolant temperature (K)*	
		Inlet	*Outlet*	*Inlet*	*Outlet*
1	*323*	*307*	*303*	*323*	*316*
2	*328*	*306*	*304*	*326*	*313*
3	*333*	*308*	*305*	*321*	*313*
4	*338*	*310*	*307*	*337*	*328*

The table 3 shows the results obtained by the experimental analysis.

All rights reserved by *www.grdjournals.com*

B. CFD Results

1) Contours for Different Orientation of Radiator Tube

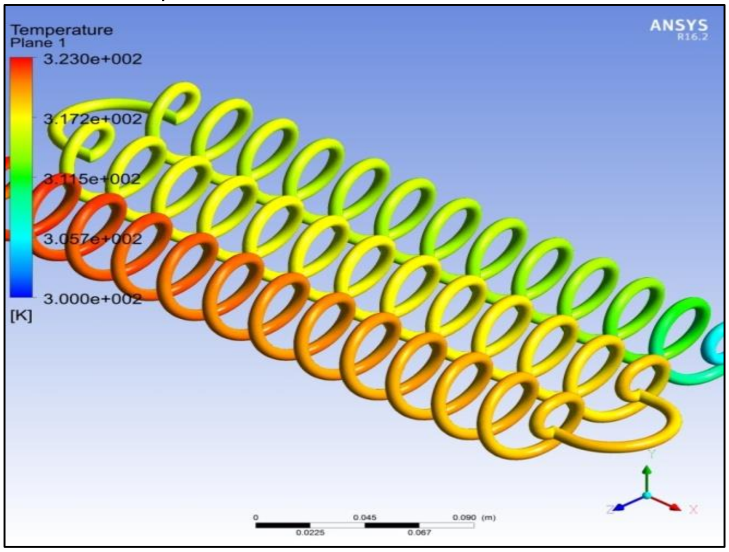

Fig. 4: Temperature contour for horizontal tube

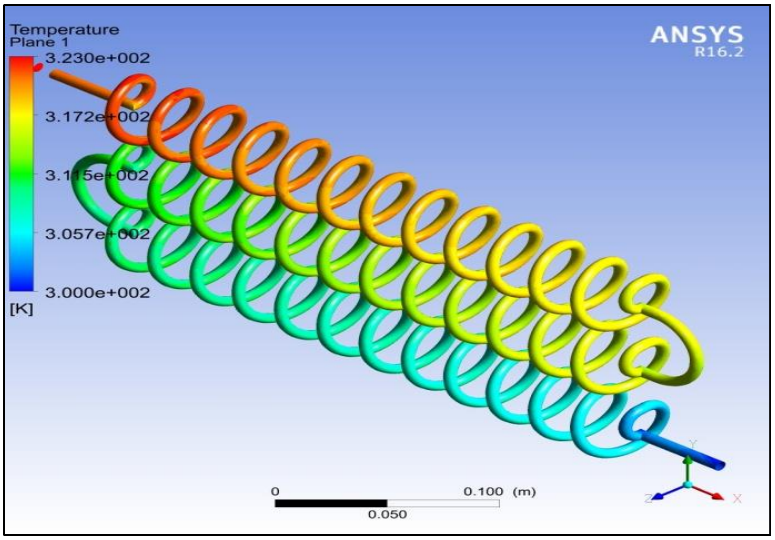

Fig. 5: Temperature contour for vertical tube

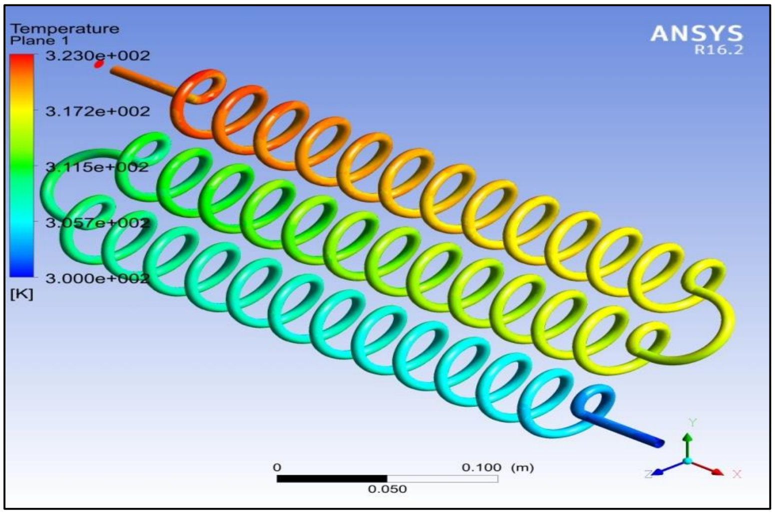

Fig. 6: Temperature contours for inclined orientation

All rights reserved by *www.grdjournals.com*

IV. CONCLUSION

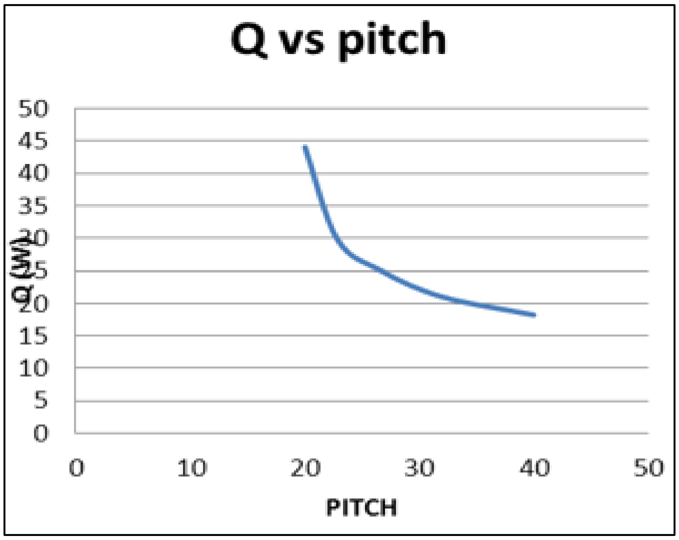

Fig. 7: Variation of heat transfer rate with pitch

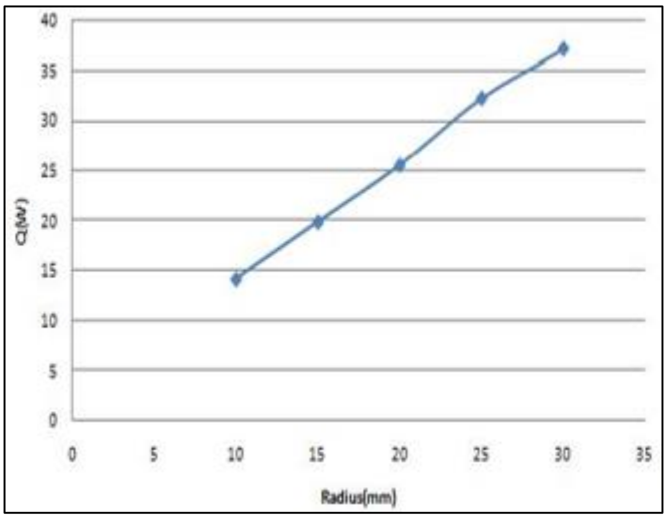

Fig. 8: Variation of heat transfer rate with radius

From the CFD simulation details we obtain the graph and its shows when the pitch increases the heat transfer rate will decreases. This graph shows the when the radius increases the rate of heat transfer also increase.

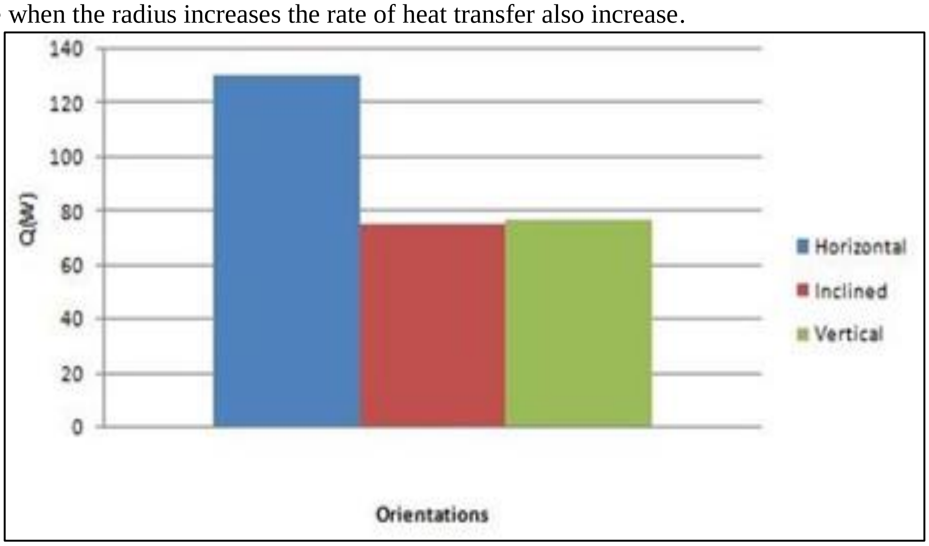

Fig. 9: Heat transfer rate for various orientations

All rights reserved by *www.grdjournals.com*

This graph shows the heat transfer rate and the different variation. The CFD results are validated using experiment. It is observed that outlet water temperature from CFD simulation is 42.076 0 C and that from experiment result is 430 C. Thus there is only a variation of 2.15% between both the results. From this analysis the maximum heat transfer is obtained for the tubes arranged in horizontal orientation which is 42% higher than other configuration When the pitch of the helical tube decreases, heat transfer rate increases by 59 %When the diameter of the helical tube is increased, heat transfer rate also increased by 61%. The horizontal orientation is compared to the vertical and inclined arrangement it have some increase in the heat transfer rate.

REFERENCES

[1] C. D. Pérez-Segarra, C. Oliet & A. Oliva (2008), Thermal and Fluid Dynamic Simulation of Automotive Fin-and-Tube Heat Exchangers, Part 1: Mathematical Model, Heat Transfer Engineering, 29, 484-494
[2] Datta N. Mehtre&Sandeep S. Kore (2014), Experimental Analysis of Heat Transfer from Car Radiator Using Nano fluids,International journal of Mechanical engineering and computer applications, 2, 101-106
[3] D. Bohne, S. Fischer, and E. Obermeier (1984), Thermal Conductivity, Density, Viscosity, and Prandtl-Numbers of Ethylene Glycol-Water Mixtures,Ber. Bunsenges. Phys. Chem. 88, 742-744
[4] Ghanbarali Sheikhzadeh, Mohammadhadi Hajilou & Hamed Jafarian (2014), Analysis of thermal performance of a car radiator employing, Nano fluid International Journal of Mechanical Engineering and Applications, 2, 47-51
[5] G.N. Tiwari, Sanjeev Kumar & Om Prakash (2003), Study of heat and mass transfer from sugarcane juice for evaporation,Desalination, 159, 81-96.

All rights reserved by *www.grdjournals.com*