CFD ANALYSIS ON TUBE WITH DIFFERENT INTERNAL FIN PROFILES

SHAIK HIMAMVALLI, M.Tech,
CAD/CAM, A1 Global Institute of Engineering & Technology Arimadugu (Village), Markapur(M), Prakasam(Dist) -523316, A.P.

DR.CH.SREEDHAR Ph.D
Principal**,** A1 Global Institute of Engineering & Technology Darimadugu (Village), Markapur (M), Prakasam (Dist)-523316. A.P.

ABSTRACT

The heat transfer rate to a fluid flowing in pipe can be enhanced by the use of internal fins. This thesis concerned with computer simulation study of vertical tube with helical fins used to enhance their heat transfer performance subjected to natural convection heat transfer. All the main parameters which can significantly influence the heat transfer performance of finned tube have been analyzed. This project is about a thorough study of natural convection from a heated pipe having fins of various configurations using ANSYS WORKBENCH version 13.0. The material under consideration is aluminum and the free stream fluid is air. The heat transfer rate from the fins, outer wall and the overall heat transfer rate has been calculated and compared for various fin configurations. Also the surface nusselt number and surface overall heat transfer co-efficient has been found out. Temperature contours for various fin configuration has been plotted showing the convection loops formed around the heated pipe surface. Velocity contours for various fin configurations has been plotted and the motion of heated fluid is shown. Plots for nusselt number and heat transfer co-efficient are also shown. The assumptions during the analysis have been taken considering the manufacturing and practical applications and working conditions. Hence the results obtained can be referred to while solving any such kind of problems in the practical field where only natural convection is under consideration.

INTRODUCTION

The science of heat transfer is concerned with the generation, use, exchange, and conversion of heat and thermal energy between physical systems. Heat transfer is the discipline of thermal engineering that concerns the calculation of rate at which heat flows within the medium, across the interface or from one surface to another. There are different modes of heat transfer which includes:

a. Heat transfer through conduction

b. Heat transfer through convection

c. Heat transfer through radiation

CONVECTION

Convection is a process which involves mass movement of fluids. Natural convection occurs due to temperature difference which produces the density difference which results in mass movement, this process is called natural or free convection. For example, assume a plate which is maintained isothermal at temperature and the surrounding temperature is . On getting heated, the fluid near the wall moves up due to the effect of buoyancy and this hot fluid is replaced by cold fluid moving towards the wall. Hence a circular current is set up due to density difference. There is a boundary layer adjacent to the plate where the velocity and temperature and velocity vary from plate to free stream. Initially the velocity increase with increasing distance from the surface and reaches a maximum and then decrease to approach zero value. This is because of action of viscosity diminishes rapidly with distance from plate, while density difference decreases more slowly. The used of heat transfer enhancement has become widespread during the last so many years.

The need of heat transfer enhancement is to reduce the size and cost of heat exchanger equipment, or increase the heat duty for a give size heat exchanger. This goal can be achieve in two ways active and passive enhancement. The active enhancement is less common because it requires addition of external power (e.g., an electromagnetic field) to cause a desired flow modification. In the passive enhancement, it consists of alteration to the heat transfer surface or incorporation of a device whose presence results in a flow field modification. The most popular enhancement is the fin.

NECESSITY OF EXTENDED SURFACE HEAT EXCHANGERS

Heat exchangers, on the basis of their constructional details, can be classified into tubular, plate-type, extended surface, and regenerative type heat exchangers. The tubular and plate-type exchangers are the primarily used surface heat exchangers, with an effectiveness below 60% in most of the cases. The surface area density of these heat exchangers is usually, less than 700 m2 /m3. In this regard, an important fact is that the thermal conductance on both sides of the heat exchanger should approximately be the same. When one stream of the flowing fluid is gas, and the other is liquid, the heat transfer surface on the gas side needs to have a much larger surface area, as it is well known that the heat transfer coefficient for gases is much lower than that for liquids. One of the most common methods to increase the surface area and compactness is to have an extended surface (fin) with an appropriate fin density as per the requirement. This addition of fins can increase the surface area by 5 to 12 times the primary surface area. These types of exchangers are termed as extended surface heat exchangers. The heat transfer coefficient of the extended surfaces may be higher or lower than that of the un-finned surfaces. The louvered fins increase both the surface area and the heat transfer coefficient, while the internal fins in a tube increase the tube surface area, but may result in a slight reduction in the heat transfer coefficient depending on the fin spacing. However, the overall thermal conductance increases due to the presence of the extended surfaces.

The heat exchanger design involves the consideration of the mechanical pumping power expended to overcome fluid friction, in addition to the consideration of the heat transfer rate. The friction power expended with high density fluids is usually less compared to the gain in the heat transfer rate; however, for low density fluids, such as gases, the pumping power is of considerable magnitude relative to the gain in the heat transfer rate. An increase in the velocity of the fluid flow increases the heat transfer rate to something less than the first power of velocity, whereas, the frictional power expenditure increases as the cube of velocity, but never less than the square of velocity. The frictional power limitations force the designers to keep the velocities moderately low. The flow velocities can be reduced by increasing the number of flow passages in the heat exchanger. It will reduce the frictional power much more, compared to the decrease in the heat transfer rate per unit of surface area. This loss of the heat transfer rate can be made up by an increase in the surface area which, in turn, also increase the frictional power, but only in the same proportion as the heat transfer surface area. This consideration also calls for extended surface heat exchangers.

In spacecraft, aircraft, and missiles, space and weight are used sparingly. It is essential that in these vehicles, on-board heat exchange duties are accomplished in equipment, that is as light and compact as possible.

The "rate equation" for the cold plate heat exchanger, which is a mathematical statement says, that the quantity of heat, Q, (Watts) transferred in a heat transfer process is equal to the product of the heat transfer coefficient, h, (W/m2K), the surface area in the cold plate, S, (m^2), and some temperature difference or driving force, ¨T, (K or °C).

$$Q = hS\Delta T$$

Thus, for a fixed heat flow and temperature driving potential, the heat flow per unit temperature difference is maximized when the h S product is maximized.

Q/ ¨T = hΔ

The design of a cold plate involves a consideration of the heat transfer between the walls of the cold plate and the circulating fluid, as well as the pumping power expended to overcome fluid friction and to move the coolant fluid through the passages within the cold plate. For a cold plate carrying a high density fluid, the friction loss is relatively small, and it is usually not the controlling factor. But when air and other gases are employed, it is not uncommon for the cold plate to quickly dissipate its allotted power. Perhaps a more attractive method of enhancing the heat transfer is to increase the surface

The foregoing considerations have led to the development of many ways, to construct heat transfer surfaces for cold plates carrying air, as the convective medium, where the surface area density is large. These surfaces are usually referred to as compact heat transfer surfaces. These surfaces make use of some form of an extended surface or fin, to augment the primary surface and form the backbone of what is to follow.

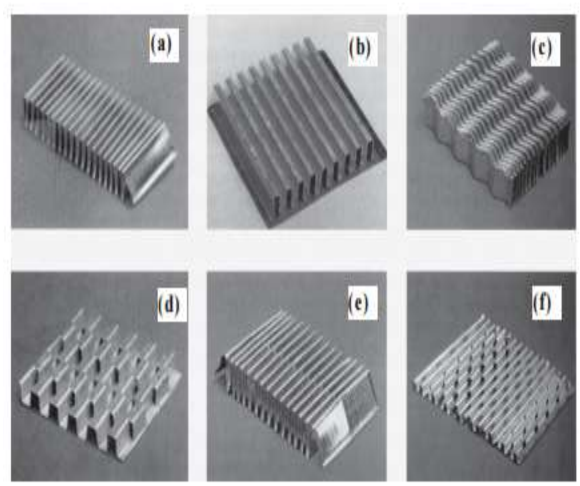


Figure: Types of Fin (a) Triangular plain fins, (b) Rectangular plain fins, (c) wavy fins, (d) offset strip fins, (e) louvered fins, and (f) perforated fins

LITERATURE SURVEY

Karthik Pooranachandran et al. "Experimental and Numerical Investigation of a Louvered Fin and Elliptical Tube Compact Heat Exchanger" carried out an experimental investigation to analyze the heat transfer characteristics of a louvered fin and elliptical tube compact heat exchanger used as a radiator in an internal combustion engine. Experiments were conducted by positioning the radiator in an open loop wind tunnel.

A total of 24 sets of air, water flow rate combinations are tested, and the temperature drops of air and water were acquired. A Numerical analysis has been carried out using fluent software (general purpose Computational fluid dynamics simulation tool) for three chosen data from the experiments. The numerical air-side temperature drop was compared with those of the experimental values. A good agreement between the experimental and numerical results validates the present computational methodology. [1]

Pega Hrnjak Et Al. "Effect of Louver Angle on Performance of Heat Exchanger with Serpentine Fins and Flat Tubes in Frosting" Measured an Effect of louver angle on performance of heat exchanger with serpentine fins and flat tubes in frosting. The results of an experimental study on the air-side pressure drop and overall heat transfer coefficient characteristics for serpentine-louvered-fin, micro-channel heat exchanger in periodic frosting. It focuses on quantification of the effects of louver angle on heat transfer and pressure drop and on defrost and refrost times. Nine heat exchangers differing in louver angle and fin pitch (i.e. louver angle 15º to 39º and fin pitch 15 to 18 fpi) were studied. The face velocity was 3.5 ms/ and inlet air relative humidity of 70% and 80%. Effect of fin pitch and louver pitch on initial Colburn j0 factor and Fanning friction f0 factor during the start of the first frosting cycle are reported, and compared to the prediction by the Chang and Wang (1997). [2]

R.Borrajo-Peláez et al. "A three-dimensional numerical study and comparison between the air side model and their/water side model of a plain fin-and-tube heat exchanger" Work based on CFD air flow models assuming constant temperature of fin-and-tube surface. The purpose of this work to present an enhanced model, whose innovation lies in considering additionally the water flow in the tubes and the conduction heat transfer through the fin and tubes, to demonstrate that the neglect of these two phenomena causes a simulation result accuracy reduction. 3-D Numerical simulations were accomplished to compare both an air side and an air/water side model. The influence of Reynolds number, fin pitch, tube diameter, fin length and fin thickness was studied. The exchanger performance was evaluated through two non-dimensional parameters: the air side Nusselt number and a friction factor. It was found that the influence of the five parameters over the mechanical and thermal efficiencies can be well reported using these non-dimensional coefficients. The results from the improved model showed more real temperature contours, with regard to those of the simplified model. Therefore, a higher accuracy of the heat transfer was achieved, yielding better predictions on the exchanger performance. [3]

Y.-G. Park and A. M. Jacobi "Air-Side Performance Characteristics of Round-

and Flat-Tube Heat Exchangers: A Literature Review, Analysis and Comparison" work on the air-side thermal-hydraulic performance of serpentine-fin, flat-tube heat exchangers. And it compared to that of conventional plate-fin, round-tube designs for various fin geometries and surface conditions. Heat exchanger performance correlations are obtained through a critical review of literature and complementary analyses. The result shows a clear advantage of lattube design under dry, low-Reynolds-number conditions in comparison to round-tube heat exchangers. The parametric effects on heat exchanger performance reported in the literature, the reasons of discrepancies, and the practical ranges of geometrical and operational parameters in applications are identified and summarized in this study. [4]

Hamid Nabati "Optimal Pin Fin Heat Exchanger Surface" represents the results of numerical study of heat transfer and pressure drop in a heat exchanger that was designed with different shape pin fins. The heat exchanger used for this research consists of a rectangular duct fitted with different shape pin fins, and was heated from the lower plate. The pin shape and the compact heat exchanger (CHE) configuration were numerically studied to maximize the heat transfer and minimize the pressure drop across the heat exchanger. A three dimensional finite volume based numerical model using FLUENT© was used to analyze the heat transfer characteristics of various pin fin heat exchangers. [5]

Pankaj N. Shrirao et al. "Convective Heat Transfer Analysis in a Circular Tube with Different Types of Internal Threads of Constant Pitch" Work on Convective Heat Transfer Analysis in a Circular Tube with Different Types of Internal Threads of Constant Pitch. This work presents an experimental study on the mean Nusselt number, friction factor and thermal enhancement factor characteristics in a circular tube with different types of Internal threads of 120 mm pitch under uniform wall heat flux boundary conditions. In the experiments, measured data are taken at Reynolds number in range of 7,000 to 14,000 with air as the test fluid. The experiments were conducted on circular tube with three different types of internal threads viz. acme, buttress and knuckle threads of constant pitch. The heat transfer and friction factor data obtained was compared with the data obtained from a plain circular tube under similar geometric and flow conditions. The variations of heat transfer and pressure loss in the form of Nusselt number (Nu) and friction factor (f) respectively is determined and depicted graphically. They observed that at all Reynolds number, the Nusselt number and thermal performance increases for a circular tube with buttress threads as compared with a circular tube with acme and knuckle threads. These because of increase in strength and intensity of vortices ejected from the buttress threads. Subsequently an empirical correlation is also formulated to match with experimental results with ± 8% and ± 9%, variation respectively for Nusselt number and friction factor. [6]

C. M. De Silva [1] et al. state that the Formula SAE vehicles, over the program's history have showcased a myriad of aerodynamic packages, each claiming specific quantitative and qualitative features. This paper attempts to critique differing aerodynamic side pod designs and their effect upon radiator heat management. Various features from inlet size, side pod shape and size, presence of an under tray, suspension cover, gills and chimneys are analyzed for their effects. Computational Fluid Dynamics (CFD) analyses are performed in the FLUENT environment, with the aid of GAMBIT meshing software and Solid Works modeling.

MATERIALS AND METHODOLOGY

INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics modeling was developed to predict the characteristics and performance of flow systems. Overall performance is predicted by breaking the flow system down into an appropriate number of finite volumes or areas, referred to as cells, and solving expressions representing the continuity, momentum, and energy equations for each cell. The process of breaking down the system domain into finite volumes or areas is known as mesh generation. The number of cells in a mesh varies depending on the level of accuracy required, the complexity of the system, and the models used. Equations solve for flow (x, y, and z velocities), energy (heat fluxes and temperatures), chemical reactions (reaction kinetics and species concentrations), and pressure based on various simplifications and/or assumptions (Anderson J. D. 1995). Some simplifications and assumptions are discussed below. If performed correctly,

CFD modeling can accurately predict the performance of an entire system.

Assumptions in CFD

The physics of conjugate heat transfer in radiator is simplified with the following technically valid 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 flow condition is characterized by the bulk speed at any cross section.

• The thermal conductivity of the solid material is constant.

• No internal source exists for thermal-energy generation

• Properties of the fluids and the wall, such as specific heat, thermal conductivity, and density are only dependent on temperature.

INTRODUCTION TO ANSYS

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

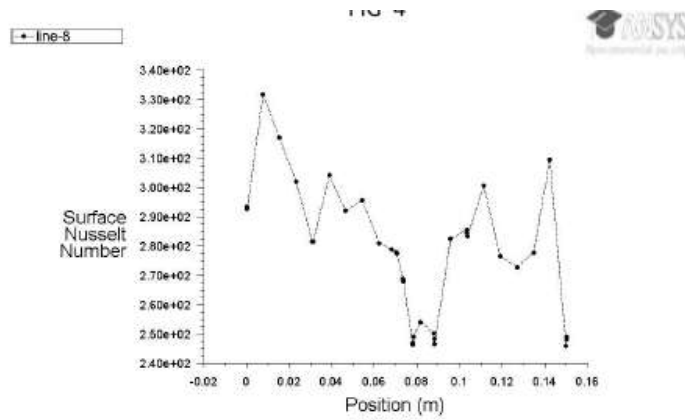
ANSYS is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. ANSYS is also used in Civil and Electrical Engineering, as well as the Physics and Chemistry departments.

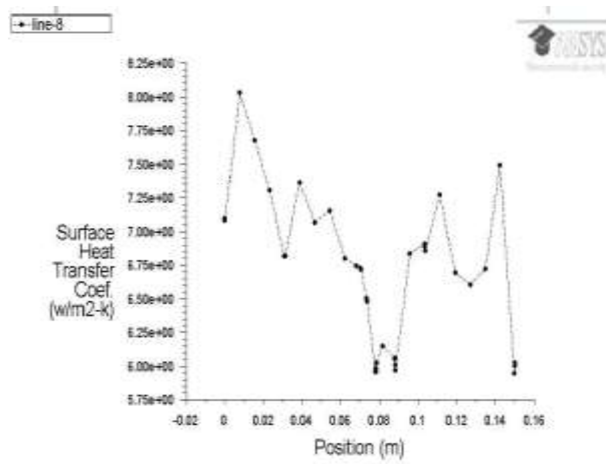
ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping.

With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of a design on the whole behavior of the product, be it electromagnetic, thermal, mechanical etc.

Figure below shows the variation of temperature across the tubes with 10 concave shaped fins internally arranged. Maximum temperature of 369K was observed

PLOT FOR NUSSELT NUMBER AND HEAT TRANSFER COEFFICIENT





Problem Data:
Length of the tube 0.2 metres.

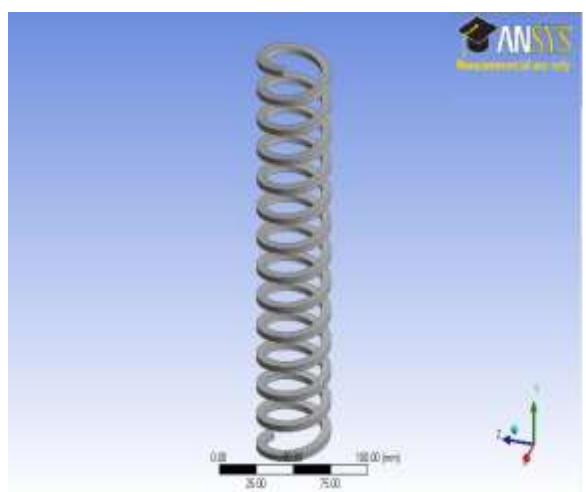


Fig: One helical fin with large number of turns

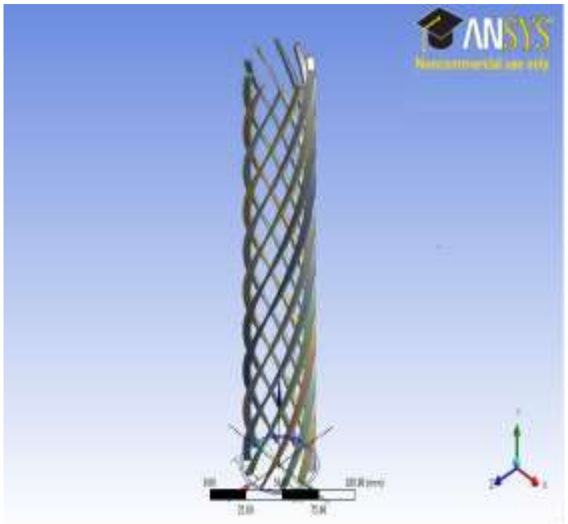


Fig. Ten helical fins with single turn

RESULTS AND DISCUSSIONS

TEMPERATURE CONTOURS FOR DIFFERENT TUBES:

The temperature counters of the vertical tube without fin, maximum temperature of 365K is observed at the wall due to convection specified and the boundary.

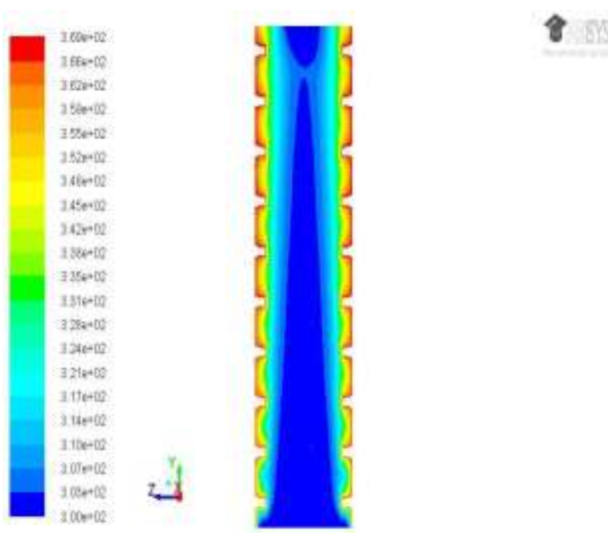


Fig:Temperature Contours of tube5 in vertical plane

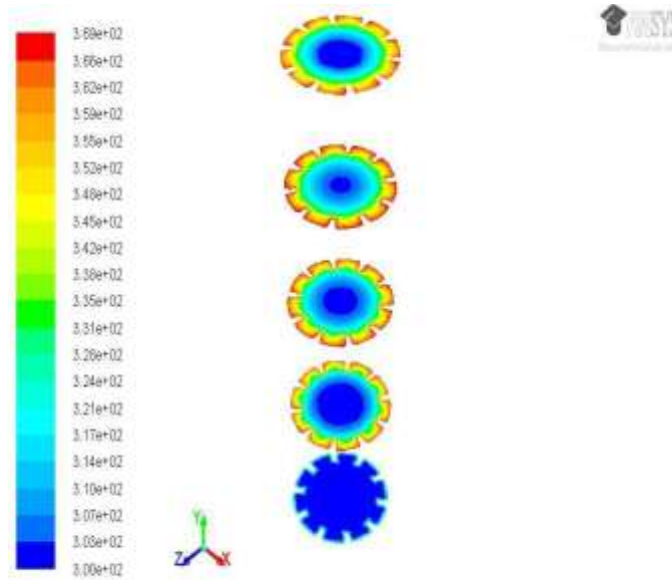


Fig: Temperature contours of tube5 in horizontal planes

Tube1:

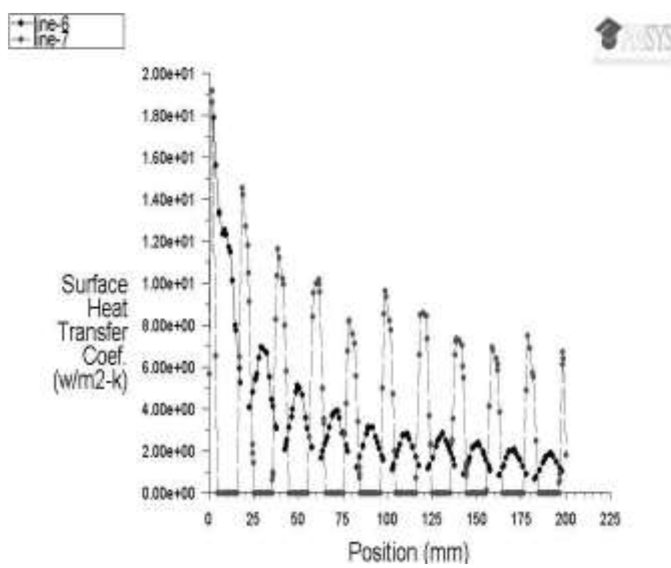


Fig: line-6 passes from fin base and line-7 passes from fin tip.

Tube:

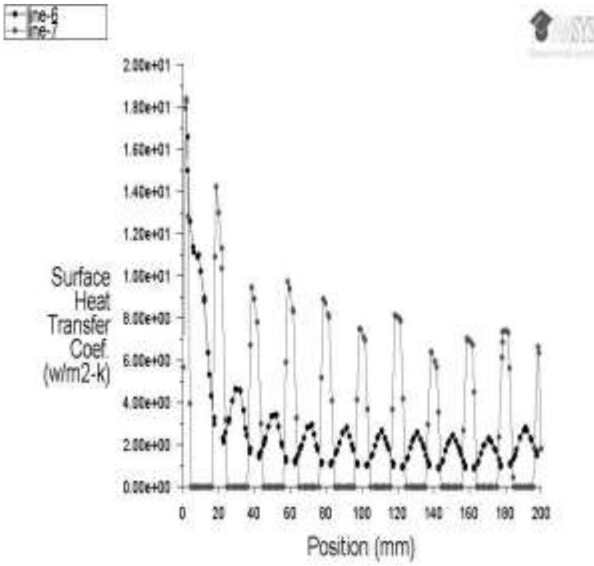


Fig: line-6 passes from fin base and line 7 passes from fin tip.

HEAT TRANSFER RATE

Table: Heat transfer rate for different tubes.

Tube no.	Heat transfer from fin(W)	Heat transfer from inner wall (w)
1	----------	11.054466
2	7.0425867	5.6044407
3	9.6723562	7.1381139
4	11.977043	6.2673212
5	11.243548	5.7177154

CONCLUSIONS

1. The tube1, tube2 and tube3 have been compared on the basis of different graphs governed from the CFD analysis and it has seen that from fig. 4.1, 4.2 and fig. 4.3, fin configuration in tube3 is more effective than other two tubes. The geometry of fins used in Tube2 has more restricted path for the air flow which increases the flow resistance and decreases the air flow rate and that downs the heat transfer rate. From fig. 4.10, fig 4.11 and fig. 4.12, it can be seen that value of surface nusselt number has maximum value for tube3 as compared to tube1 and tube2. For tube3, near the bottom point of the tube, it is more than 300 which is greater than the tube2 which has nearly equal to 250. The surface heat transfer coefficient is compared at different position on the tube, and has more value for tube3, nearly equal to 9W/ -k at the lowest position of the tube, as compared to 5.5W/ -k and 4.5W/ -k for the tube1 and tube2 respectively. Heat transfer rate is 11.05 W, 12.647 W and 16.81 W respectively for the tube1, tube2 and tube3. Tube3 has maximum heat transfer rate. Hence the results showed that, for tubes having different fin configurations, the tube having ten equally spaced internal helical fins is more effective as compared to the tube without fin and tube2 which has one helical fin with large number of turns.

2. Tube3, tube4, tube5 having same fin configuration, which already had been concluded, have been compared for best fin profile. Tube3, tube4 and tube5 have rectangular, trapezoidal and concave parabolic fin profiles respectively. From fig. 4.13, fig. 4.14 and fig. 4.15, it has seen that at the position of 20mm from the bottom point of the tube the value of surface nusselt number is 450 for tube3, for tube4 it is more

than 600 which is greater than tube5 which has less than 600. The value of surface heat transfer coefficient has approximately equal values for tube4 and tube5 of approximately equal to 14W/ -k as compared to tube3 of approximately equal to 10W/ -k. Heat transfer rate from tube4 is 18.244 W which is more than 17.061 W and 16.81 W for tube5 andtube3 respectively. Hence the overall performance of the fins and heat transfer rate from different fin profile has maximum value for trapezoidal fins for natural convection through internal fins for the given case.

REFERENCES

[1] Karthik Pooranachandran et al. " Experimental And Numerical Investigation Of A Louvered Fin And Elliptical Tube Compact Heat Exchanger", email: velrajr@gmail.com .

[2] Pega Hrnjak Et Al. "Effect Of Louver Angle On Performance Of Heat Exchanger With Serpentine Fins And Flat Tubes In Frosting", *2501*, Page 1

[3] R.Borrajo-Peláez et al. "A three-dimensional numerical study and comparison between the air side model and theair/water side model of a plain fin-and-tube heat exchanger", Applied ThermalEngineering 30 (2010) 1608-1615

[4] Y.-G. Park and A. M. Jacobi "Air-Side Performance Characteristics of Round- and Flat-Tube Heat Exchangers: A Literature Review, Analysis and Comparison", Acrccr-36, May 2001

[5] Hamid Nabati "Optimal Pin Fin Heat Exchanger Surface", 2008, School of Sustainable Development of Society and Technology, ISSN 1651-9256, ISBN 978-91-85485-95-6, Printed by Arkitektkopia, Västerås, Sweden

[6] Pankaj N. Shrirao et al. "Convective Heat Transfer Analysis in a Circular Tube with Different Types of Internal Threads of Constant Pitch"

[7] C.M.Desilva, M.Norazmi, T.Christie, E. Abou- Saba, A. "Computational Flow Modelling of Formula-Sae Side Pods for Optimum Radiator Heat Management" Journal of Engineering Science and Technology, Vol.6, No.1 (2011), 94-108.

[8] Salvio Chacko, Dr.Biswadip Shome, Vinod A. Kumar, K.Agarwal, D.R.Katkar "Numerical simulation for improving radiator efficiency by airflow optimization" Engineering Automation Group, Tata Technologies Limited, Pune, India Engineering Research Center, Tata Motors Limited, Pune, India.

[9] Pawan.S.Amrutkar, Sangram.R.Patil "Automotive radiator performance–Review" International Journal of Engineering and Advanced Technology (IJEAT)ISSN:2249– 8958,Volume-2,Issue-3,February2013

[10] Nag P.K., Heat and mass transfer, New Delhi, TMH, 2010

[11] Kayansayan N., Thermal characteristics of fin and tube heat exchanger, Experimental thermal and fluid science, 1993, Vol. 7, pg. 177-188

[12] Rao V.D., Naidu S.V., Rao B.G., Sharma K.V., Heat transfer from a horizontal fin array by natural convection and radiation, International journal of heat and mass transfer, 2006,
Vol. 49, pg. 3379-3391