RESEARCH ARTICLE OPEN ACCESS

Conjugate Heat transfer Analysis of helical fins with airfoil cross-section and its comparison with existing circular fin design for air cooled engines employing constant rectangular cross-section

Ashwin Shridhar*, Asokan Ram Deepak**, S Jayaraj***
*(Corresponding Author, Department of Mechanical Engineering, National Institute of Technology, Calicut, Kerala, INDIA 673601)
** (Department of Mechanical Engineering, National Institute of Technology, Calicut, Kerala, INDIA 673601)
***(Principal Investigator, Department of Mechanical Engineering, National Institute of Technology, Calicut, Kerala, INDIA 673601)

Abstract
Air Cooled Engines have been used in a variety of applications, ranging from airplanes to motorbikes and even stationary or portable engines. Since modern automobiles and airplanes use engines delivering more power, they have to be cooled more efficiently due to which a more complex water cooling system is used for cooling engines with large displacements. Hence air cooling is becoming a thing of the past, especially in the aviation sector due to the advent of more efficient gas turbine engines. However air cooled internal combustion engines are still being used in a wide variety of two-wheelers ranging from small single cylinder engines to heavy duty liter class V-twins and Inline fours, due to the non-practicalities associated with the installment of a bulky water cooling system in two-wheelers. So one can ascertain that there is a scope for improving the efficiency of air cooled engines even further. The objective of this paper is to analyze currently existing fin design employed in most of the air cooled engines and improve it by changing the cross-section to a streamlined one and also making the fins in a helical orientation as opposed to the regular circular fins employed. Our analysis comprises of a computational fluid dynamics study of both the fin models with identical dimensions and simulated in the same environment using ANSYS FLUENT 15 software and we attempt to compare their performance using the temperature and heat transfer coefficient distribution plots obtained.
***Keywords* - — Airfoil, Computational Fluid Dynamics (CFD), Conjugate Heat Transfer analysis, Fins, Heat Transfer, Simulation**

I. INTRODUCTION

Air Cooling is one of the oldest and effective forms of engine cooling techniques used mainly in airplanes and small automobiles like two-wheelers. However, there has been an increasing demand for high efficiency and high specific power output engines which necessitates a detailed study of engine subsystems of which cooling system is an important component to consider. Air cooled motorcycle engines release heat to the atmosphere through forced convection where air flows over the fins provided on the engine cylinder. It is the airflow that does the work of keeping the engine cool by absorbing the heat energy from the fin surface, so an air-cooling system is usually simple both in theory and practice. This aspect is useful for an engine where weight minimization is an important factor and since the heat transfer is directly proportional to the velocity of the flow, air cooling works best on the engine that's exposed to a high airflow environment. Also the velocity distribution over the fin surface is of paramount importance. Our work is based on the experiments conducted by Thornhill (et.al 1) and the CFD study done by Agarwal (et.al 2). The basic dimensions of our finned cylinder model is also based on the one used by Thornhill in his experiment. The performance of an air-cooled engine depends on various fixed and variable parameters viz. flow velocity, ambient temperature, fin geometry material etc. Our work is an attempt to improve the performance by changing the geometry of the fin while keeping other parameters constant. We have changed the cross-section of the fin to a streamlined one in an attempt to increase the flow velocity over the fin surface and hence facilitating more heat dissipation due to a proportional increase in heat transfer coefficient. In addition to the cross-section change, the fin was modelled over the cylinder in a helical orientation so as to accommodate more fin area than the conventional circular finned model, over the cylinder with same dimensions. We also believe that this helical arrangement results in the forming of a smaller wake region than the circular fin, hence improving cooling at the back side of the engine. Alongside the CFD analysis we have also formulated a generalized differential equation pertaining to the energy balance of the helical finned model which can be solved using numerical techniques to obtain the temperature distribution theoretically.

II. Methodology

(A) Fin design and Modelling

Thornhill (et.al 3) has performed a similar study on cooling of finned metal cylinders where he kept the shape of the fins as circular of constant thickness with inner diameter of the cylinder as 100mm and varied fin length from 10 to 50 mm. He also varied the fin pitch from 8 to 14 mm and used a wind speed in the range of 7.2 to 72 km/hr. We have adopted the above design for our calculations keeping most of the dimensions unaltered save for the cylinder thickness, fin pitch and the fin thickness due to design constraints for the helical model (see Figure 1 and 2). Table 1 and Table 2 show the geometric parameters for the circular and helical fin model respectively.

Our helical fin model is based on the above design except the cross section of the fin, which is an airfoil instead of the one with a constant thickness as mentioned above. The airfoil is a NACA64012a with a symmetric profile. The primary airfoil dimension viz. the chord length was chosen appropriately to closely follow the dimensions adopted from Thornhill. A fillet of radii 2mm was provided at the base of the airfoil fin to facilitate its manufacturing and smooth execution of the solution by the FLUENT solver, no fillet radii was provided for the circular fin model. Both the fin models were created using Creo Parametric 2.0 Software and were imported into FLUENT software for performing the CFD calculations. The material for the model was chosen to be aluminum which is the most common material employed for making engine fins.

Fig. 1 Circular finned model used for the experiment

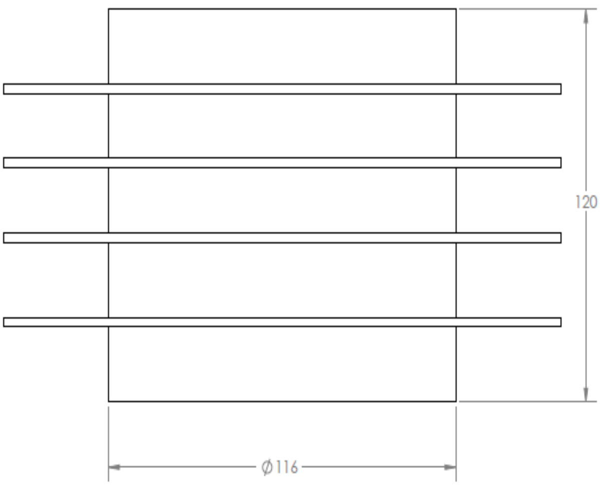


TABLE 1 Geometric parameters for circular finned model

Parameter	**Dimensions**
Cylinder Inner diameter	100mm
Cylinder thickness	16mm
Fin thickness	3mm
Fin Pitch	19.8 mm
Fin length	40mm
Total Length of the cylinder	120mm

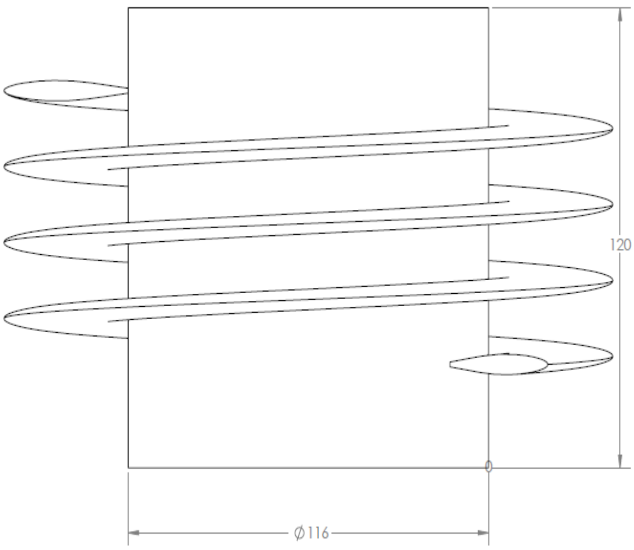


Fig. 2 Helical fin with airfoil cross section model used for the experiment

Parameter	Dimensions
Cylinder Inner diameter	100mm
Cylinder thickness	16mm
Airfoil specification	NACA64012a
Chord Length	45mm
Fin Length	40mm
Fin Pitch	19.8mm
Total Length of the Cylinder	120mm
Fillet Radii	2mm

TABLE 2 Geometric Parameters for helical finned model

(B) Mathematical modelling

We have attempted to formulate a generalized differential equation for the energy balance of the helical fin model. This equation is based on the differential equation for circular fins given in Cengel (et.al 4) and a similar cylindrical coordinate system was employed for the derivation. In order to derive the equation, we replaced the circular differential element with a helical element. The differential equation hence obtained can be given as

$$\frac{1}{\sqrt{r^2+b^2}}\frac{\partial}{\partial r}\left(t\sqrt{r^2+b^2}\frac{\partial T}{\partial r}\right)+\frac{1}{r^2+b^2}\frac{\partial}{\partial \theta}\left(t\frac{\partial T}{\partial \theta}\right)-\frac{2h}{k}(T-T_\infty)=0 \quad (1$$

Where

r = radial distance from wall of cylinder to edge of fin

$2\pi b$ = pitch

t = thickness

T = temperature

h = Convection Heat transfer coefficient in W/m^2K

k = Conduction Heat Transfer coefficient in W/m^2

T_∞ = Free stream temperature

One can deduce the fact that the above equation reduces to the one depicted in [4] for circular fins when the pitch parameter b is set to zero, which supports the aptness of this equation for defining the energy balance occurring in the helical fin model.

The solver employed by FLUENT uses the generalized three dimensional form of navier-stokes equation and the energy equation (Equations 2 and 3 respectively) for obtaining the fluid flow and heat transfer parameters respectively. Since these equations are generalized they were directly adopted from the paper published by Agarwal. They are respectively given as

$$\frac{\partial(\rho v)}{\partial t}+v\,\nabla.(\rho v) = -\nabla P+\nabla.\tau+\mathrm{F}+\rho g \qquad (2)$$

$$\frac{\partial(\rho E)}{\partial t}+\nabla.(v(\rho E+p)) = \nabla.\left(k_{eff}\nabla T-\textstyle\sum_j h_j J_j+(\tau.v)\right)+S_h \qquad (3)$$

In addition, steady state condition was assumed and the Spalart-Allmaras model was used which is preferred for external flows.

(C) Simulation Analysis

The conjugate heat transfer study was performed on a laptop computer with 8 GB Ram and Intel® Core™ i7 CPU @ 3.1 GHz running on 64 bit Windows 8 Operating system using ANSYS FLUENT 15.0 software. The geometry of the finned cylinders were modelled using CREO Parametric 2.0 and were meshed with a tetrahedral element in FLUENT using a custom body sizing mesh control with a minimum element size of 2.5mm and the local maximum element size was kept as the default value for the coarse setting and the growth rate was kept unaltered. This size range was used for a fast as well as an accurate solution to our problem. The air speed was assigned as 20m/s (72km/hr.) which is the upper limit used by Thornhill. The ambient air temperature was assigned as 300 K and outlet pressure was assigned as atmospheric pressure i.e. 101.325KPa. The air was assigned an Ideal-Incompressible gas model so as to accommodate the slight density variations encountered during the flow and hence maintaining the validity of the equations used for incompressible flows. The temperature inside the finned cylinder was kept as 473 K to account for the heat generated during the combustion process.

An enclosure was created around the finned geometry to simulate the region of airflow. The inlet face of the enclosure was given a velocity boundary condition and the outlet face was given a pressure boundary condition so as to maintain continuity of the flow, hence simulating the movement of a body with subsonic speed in the atmosphere. All other faces of the enclosure were given an adiabatic boundary condition. The energy equation was enabled and the Spalart-Allmaras model was used to solve for the distributions. This model is a one equation model which solves for Turbulent Kinematic Viscosity. The values of all the parameters in the model were kept unaltered. The solution scheme was kept as SIMPLE and the solution was computed keeping the airflow inlet boundary as reference. All other settings were kept unaltered. The number of iterations were limited to 1000 as any value above this produced insignificant differences in the solution.

III. Result and Discussions

(A) Velocity Distribution

The Velocity distribution for the circular and helical fins are shown in Figure 3 and 4. From the contour plot one can discern that the maximum velocity reached for the helical fin model is greater than that of the circular fin model due to the employment of streamline cross-section. The flow takes place from right to left and there is wake formation due to flow separation at the leeward end of the cylinder in both the models, but one can see that the wake region in the helical fin model is less than that of the circular fin model due to its helical arrangement.

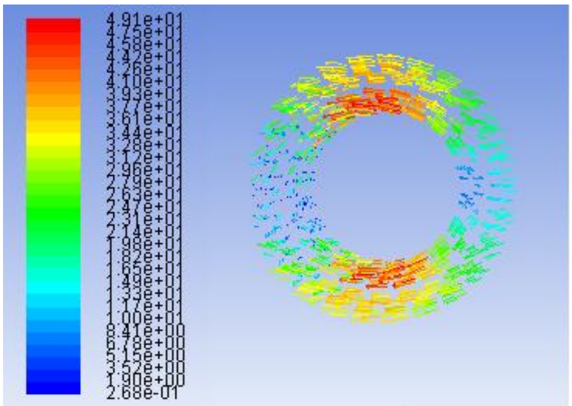


Fig. 3 Circular Fin – Velocity contour plot

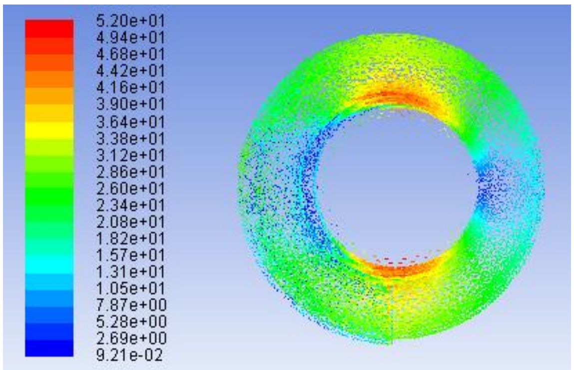


Fig. 4 Helical Fin – Velocity contour plot

(B) Temperature distribution

The temperature contour plot was obtained for both the models (See Figures 5 through 8). As mentioned previously the flow is directed from right to left and due to wake formation the temperature drop at the backside of the cylinder is less compared to the front and sides. We have also obtained a scatter plot of the temperature distribution varied with the angular coordinate of the cylindrical coordinate system we previously mentioned. From these plots we can see that although the minimum temperature attained in both the models are almost same, the temperature variation is smoother in case of the helical fin model, though the minimum temperature for the helical fin model unusually doesn't occur at the sides of the cylinder due to the introduction of fillet radius at the base part of the fin, so we can say that the fillet radii largely determines the temperature distribution around the fin and it must be set to a minimum value to facilitate a better airflow over the fin surface. But we can safely say that more points on the helical fin model are at a lower temperature compared to the circular fin model hence from the temperature plot one can say that the helical fins with airfoil cross-section are more efficient in heat dissipation.

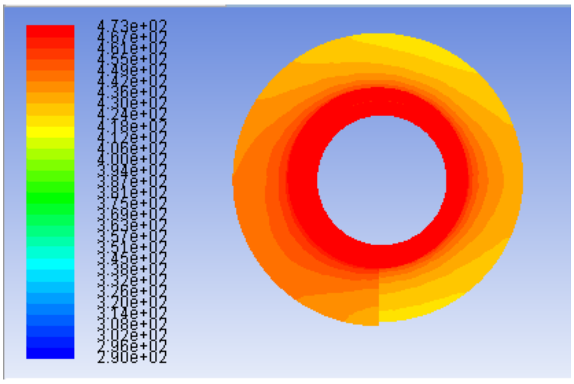


Fig. 5 Helical Fin – Temperature Variation

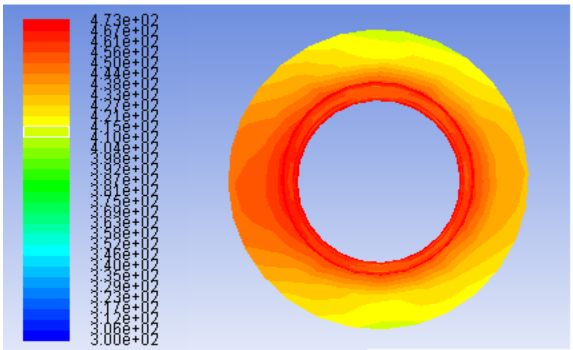


Fig. 6 Circular Fin – Temperature Variation

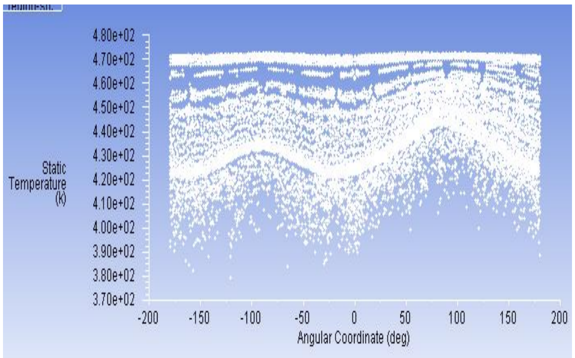


Fig. 7 Helical Fin – Temperature Variation with Angular Coordinate

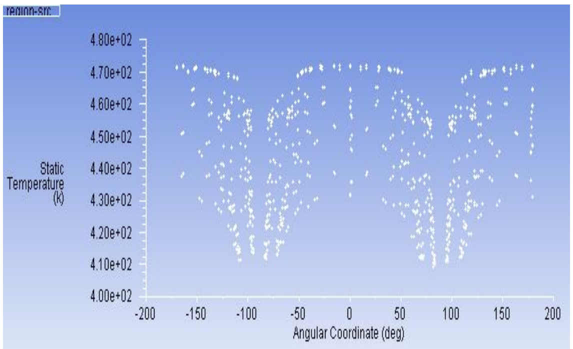


Fig. 8 Circular Fin – Temperature Variation with Angular Coordinate

(C) Heat Transfer coefficient Distribution

The Wall function heat transfer coefficient contour plots were obtained for both the models (see Figures 9 through 12). Since its value directly depends on the local velocity at that point, we can very well correlate the values obtained in this plot with the values of the velocity distribution. We have also obtained a scatter plot depicting the variation of heat transfer coefficient distribution along the angular coordinate for both the models. From these plots we can observe that the maximum value of the heat transfer coefficient attained for helical fin model is greater than that of circular fin model, which can be explained by the velocity distribution over both the models. Also we can make an unusual observation for the helical fin model that the heat transfer coefficient is not maximum at the sides like that of the circular fin model, this is due to the

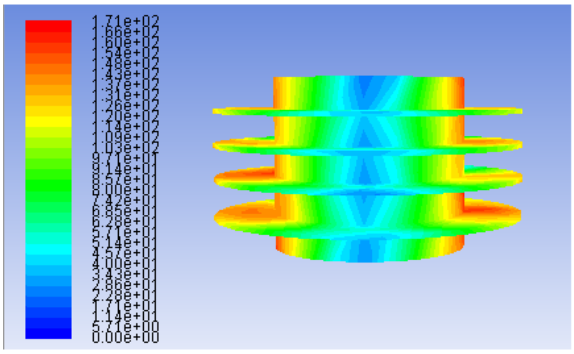
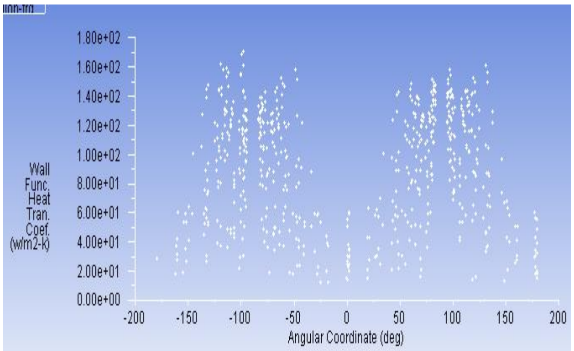


Fig. 9 Circular Fin – Heat Transfer Coefficient (Contour Plot)

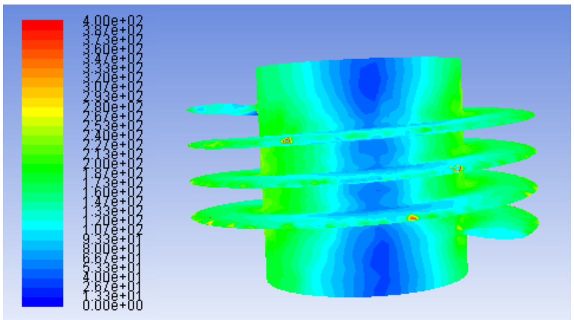


introduction of a fillet radii which tend to disrupt airflow near the base. As mentioned in previous section the fillet radii should be minimized so as to keep a reasonable efficiency for the helical airfoil fin.

Fig. 12 Helical Fin – Heat Transfer coefficient (Scatter Plot)

Fig. 11 Helical Fin – Heat Transfer Coefficient (Contour Plot)

Fig. 10 Circular Fin – Heat transfer coefficient (Scatter Plot)



IV. Conclusion

The significant conclusions derived from our analysis are depicted below

1. We have successfully compared our helical airfoil fin model with a standard rectangular cross-section circular finned model by performing a Conjugate Heat transfer analysis on both models in the same environment.
2. From the CFD analysis we have obtained the Velocity distribution, Temperature distribution and the heat transfer coefficient distribution for both the models in the form of contour and scatter plots.
3. By observing the aforementioned distributions we came to a conclusion that the helical airfoil model is more efficient than the circular fin model with rectangular cross-section.

REFERENCES

[1] D. Thornhill D, A. Graham, G. Cunningham, P. Troxier and R. Meyer, “Experimental Investigation into the Free Air-Cooling of Air-Cooled Cylinders”, SAE Paper 2003-32-0034, (2003).
[2] Pulkit Agarwal, Mayur Shrikhande and P. Srinivasan, “Heat Transfer Simulation by CFD from Fins of an Air Cooled Motorcycle Engine under Varying Climatic Conditions”, World Congress on Engineering 2011 Vol III WCE 2011, July 6 - 8, 2011, London, U.K.
[3] D. Thornhill and A. May, “An Experimental Investigation into the Cooling of Finned Metal Cylinders in a free Air Stream”, SAE Paper 1999-01-3307, (1999).
[4] Yunus Cengel, Heat Transfer, (2nd Edition, McGraw Hill (Tx), 2002)