Numerical investigation on air-cooling enhancement of a motor cycle engine by varying fins geometry

Bibhuti B Sahoo\textsuperscript{1,4}, Chandrakanta Nayak\textsuperscript{2} and Mukul Shrivastava\textsuperscript{3}

\textsuperscript{1}School of Mechanical Engineering, Vellore Institute of Technology, Vellore – 632014, India, \texttt{bbs.iitg@gmail.com}
\textsuperscript{2}Department of Mechanical Engineering, Gandhi Institute of Technological Advancement, Bhubaneswar – 752054, India, \texttt{chandra_kec@rediffmail.com}
\textsuperscript{3}Department of Mechanical Engineering, Vardhaman College of Engineering, Hyderabad – 501218, India, \texttt{mukul@vardhaman.org}
\textsuperscript{4}Corresponding author

Abstract. Two-wheeler vehicles are equipped with an air-cooled cooling system. Internal combustion engines generate heat to obtain mechanical energy but non-convertible heat energy is dissipated from engine heat and walls. Air-cooled fins provide extra material with an extended surface area to enhance heat transfer rate. The performance of fins can be increased by changing different parameters like material, geometry, number of fins etc. Present work combines the changes in the fin geometry, shape, and pitch to maximize the heat transfer rate as compared to earlier available works. Modelling of 100 cc engine with cooling fins is recreated using CAD software SOLIDWORKS. A heat thermal analysis of original and modified models was done with Mechanical ANSYS and Fluent. In the thermal analysis, temperature variation along the length of the fins was analyzed. The modified model was able to achieve an 8.5\% increase in heat transfer rate with air-motion as compared to the existing original 100 cc engine cylinder.

Keywords: Two wheeler, Cooling fins, Heat transfer, Modelling, Air-cooling

1. Introduction and background
The reciprocating internal combustion (IC) engines produce high temperature and pressure during the combustion of fuel. This force imparts on the piston in the expansion stroke which produces mechanical energy. The melting point of the engine cylinder material is generally lower than this high IC engine combustion temperature. Hence, practically, the cylinder temperature is reduced by some cooling methods to avoid a complete seizure of the piston, bearing and other important parts. Also, higher engine temperatures decrease engine performance by lowering the volumetric efficiency, promoting pre-ignition and detonation. Hence, IC engines are deliberately designed for heat losses of approximately 70\% through cooling systems to prevent the thermal damage of cylinder material. Only about 30\% of engine heat is converted for useful mechanical power [1, 2]. The air-cooling method is mainly adopted in small capacity IC motor bike engines effectively due to space constraints and sufficient air flow availability. In this method, the extended surfaces from the combustion chamber called ‘fins’ are used to cool the engine. Air flowing over and around the cylinder carried away engine...
heat through convective heat transfer. Fins are cast on the cylinder block and head to increase the heat transfer rate by increasing the convective surface area. The amount of engine heat to be dissipated through fins depends on some factors such as geometry, shape, size, material, wind speed, etc. Some numerical and experimental studies were done by the researchers to optimize these fin factors to enhance the engine heat transfer rate [3-8].

In experimental work, the elliptical fin was found to be more efficient in transferring engine heat over a circular fin [9]. The gap between two consecutive fins was concluded as the main parameter for engine heat dissipation rather than fin quantity in experimental work by Yosidha et al. [10]. When the ambient temperature is reduced to a very low value, the small engines are usually overcooled and the fin performance increases with an increase in ambient temperature [1, 11]. In another experimental work, when fin geometry was changed from regular rectangular to porous fins, it produced a 5.6% higher heat transfer rate [12]. The fins surface with holes or perforation fins reduces the materials use and also, increases the heat transfer rate from the engine. It was confirmed in a motor bike modelling work where the researcher found 5% to 6% increase in heat transfer rate for holed fins design [2, 13-14]. Due to the decrease in Nusselt number, it was found in numerical analysis that the permeable fins produced significant heat rate enhancement over the solid fins [15]. Aluminum fins produce a high heat transfer rate as compared to copper fins. But, the copper material is more stable at the lowest temperature [16].

Fin configuration is one of the important parameters which decides the heat transfer rate [17-19]. Using software simulation of an air-cooled engine, a higher heat transfer rate was achieved for perpendicular aligned circular shaped fins instead of rectangular shaped fins and fin materials [20]. In another simulation work using ANSYS software, the trapezium shape fins were produced 4% to 13% more heat transfer rate than the triangle, rectangle and circular fins [21]. Researchers have also found that convex and conical shaped fins are more efficient in heat transfer from the slam air-cooled engine [22-23].

The previous research works on air-cooled fin for the IC engine have demonstrated the enhanced heat transfer rate through varying on the fin parameters such as material, geometry and shape, surface area, perforation, etc. individually. Hence, to enhance the heat transfer rate to a maximum level, a combination of fin parameter variation can be adopted on the air-cooled fins. To achieve this objective in this work, we have combined the changes in the fin geometry, shape, and pitch of a 100 cc motor bike engine through 3D modelling SOLIDWORKS. Subsequently, its heat transfer performance was analysed and compared through ANSYS mechanical and FLUENT software.

2. Methodology

2.1. CAD model

The two-wheeler engine taken here for the study was a 100 cc capacity Honda Make motor cycle. The complete dimensions of air-cooling arrangement of this engine considered for CAD modelling is presented in Table 1. The original design of air-cooling was modified by changing different parameters of fins, such as shape, surface and pitch. The modified fin arrangement with the engine cylinder is also modelled as per the dimensions given in Table 1. The CAD modelled designs are presented in Figures 1 and 2.

2.2. ANSYS simulation

The CAD models were imported to ANSYS workbench for the heat transfer analysis with help of APDL 16.0. The ANSYS software is provided with simple meshing methods where node sizes can be varied according to need in the same part with a varying shape with internal temperature 300ºC and convection coefficient of air. Theoretical methods were used in the ANSYS simulation of models without air flow. The surface area of the original and modified model was calculated as 0.13565436 m² and 0.135877.5 m² respectively. Whereas, the total heat flux over the area of the original and modified model was calculated as 6328.2 W/m² and 6579.4 W/m² respectively.
Table 1. Dimensions Details of CAD Models

| Fin Parameter          | Original Design | Modified Design |
|------------------------|-----------------|-----------------|
| Number of fins         | 30              | 45              |
| Pitch                  | 7.5             | 3.5             |
| Surface area           | 1350.60 cm²     | 1821.18 cm²     |
| Volume                 | 330.98 cm³      | 373.05 cm³      |
| Number of holes        | 0               | 12/fin          |
| Fin end shape          | Rectangular     | Triangular      |
| Extended surface       | No              | Trapezoidal     |
| Area/ volume           | 4.1             | 4.9             |

Figure 1. Original model of engine  
Figure 2. Modified model of engine

Method 1: Theoretical method for temperature distribution data obtained from ANSYS simulation.

Heat transfer from fins, 

\[ Q = \sqrt{hpKA} \times \theta \times \frac{[\tanh(ml) + (hl/mK)]}{1 + [(hl/mk) \times \tanh(ml)]} \]

where, \( \theta = (T_e - T_i) \)

\( p = \) perimeter (m),
\( l = \) length (m),
\( A = \) surface area (m²),
\( m = \sqrt{(hp/KA)} \)

\( h = \) convective heat transfer coefficient (W/m²K), and
\( K = \) thermal conductivity (W/mK).

The temperature gradient for both models using the above mathematical relations is presented in Figures 3 and 4.
Figure 3. Temperature gradient of original model

Figure 4. Temperature gradient of modified model

Method 2: Calculating total heat flux over the outer surface area of the fin model.

\[ \text{Flux of } F \text{ through } S = \int \int_S (F \cdot n) dS \]

Where, \( F \) = heat flux (W/m\(^2\)), \( S \) = surface area (m\(^2\)), \( n \) = normal direction vector.

Figure 5. Heat flux gradient of the original model

Figure 6. Heat flux gradient of the modified model

2.3. ANSYS Fluent analysis

The CAD model was imported into ANSYS Fluent as shown in Figure 7. Smart meshing is done on the model can be seen in Figure 8. Boundary conditions were applied to the model as shown in Figure 9. The inner engine surface temperature was assumed for this study to be constant at 300 °C. Air velocity was taken as 45 km/hr with 22 °C ambient temperature. Total heat flux was calculated over the outside surface area as shown in Figure 10.

Surface integration methods were used along the surface, unlike normal integration methods. Surface integrals were already present independent of each other but scattered over the given surface area. Heat integrals are vector quantities that can be surface integrated over a given surface area to calculate total heat flux over the required amount of area of the body of any shape. The air Mass flow rate was calculated near the fin surface area. The final FLUENT ANSYS reports both original and modified models were generated from the analysis and presented in Figures 11 and 12 respectively.
3. Results and discussion
The complete results of the numerical analysis done on both original and modified engine models by using ANSYS Mechanical and Fluent software are presented in Table 2. Considering the surface area and total heat flux over the area of each model, the ANSYS Mechanical analysis showed that the amount of heat transfer from the original and modified engine model was 858.44 J and 894.32 J.
respectively. There is an increase of 4.18% heat transfer from fins with the modified model when air flow over the fins was not taken into account.

When the simulation was completed using ANSYS Fluent it was possible to consider the engine in a moving state. Here, the air flow over the fins plays an important role in increasing the heat transfer rate. It was found that the heat transfer rate from the original model was 872.32 J increased to 956.54 J with the modified model which is an increase of 8.51%. Using ANSYS Fluent analysis, it was found that the rise in air flow which causes a 4% rise in the coefficient of heat transfer is directly proportional to the heat transfer rate.

| ANSYS Software | Model      | Total heat flux (W/m²) | Net heat transfer rate (J) | Mass air flow at fin wall (kg/s) |
|----------------|------------|------------------------|--------------------------|---------------------------------|
| Mechanical     | Original   | 67411                  | 858.44                   | ----                            |
| Mechanical     | Modified   | 68121                  | 894.32                   | ----                            |
| Fluent         | Original   | 67812                  | 872.32                   | 0.208997                        |
| Fluent         | Modified   | 73581                  | 956.54                   | 0.228781                        |

4. Conclusions
The following conclusion was made from the numerical investigation on heat transfer from air-cooled motor cycle engine through fins.

i. It is validated through the above research work that the use of more than one fin design modifications techniques helps to improve the comparative heat transfer rate than the original engine head.

ii. Net change in the heat transfer rate of 3% to 4% was observed in all previous research works when the individual modification method was carried out. But, after adopting combined fins modification techniques, we observed a net rise of 8.5% in heat transfer rate is observed which is significant.

iii. The reasons for the rise in heat transfer rate achieved in the current modified model due to the following parameters as given below.

- **Area:** Use of trapezoidal extensions on the fin body increases the surface area for the same available volume.

- **Shape:** Spike shapes always enhances the heat transfer rate. So, the end shape of the fin was designed as triangular shapes instead of blunt faces.

- **Pitch:** Reducing the pitch to 3.5 mm, we were able to include more number of fins for the same surface area which directly increased the heat transfer rate.

- **Holes:** A rise in air flow along with an increase in surface area was achieved when holes were made in the fins.

- **Air velocity:** Velocity of air and convection coefficient rise and hence, higher heat transfer rate due to the turbulence in air flow was created due to less fin pitch and holes.

Acknowledgments
The authors wish to thank and acknowledge their B.Tech. and M.Tech students, Vellore Institute of Technology Vellore for their assistance towards the numerical analysis.
References

[1] Pulkit A, Mayur S and Srinivasan P Proceedings of the World Congress on Engineering 6-8 July London UK.

[2] Thornhill D and May A 1999 SAE Paper 1999-01-3307.

[3] Rathod HD, Modi AJ and Rathod PP International Journal of Mechanical Engineering and Technology (IJMET) 4(2) 328-333.

[4] Raja Rao PN and Vishnu Vardhan T 2013 International Journal of Engineering and Technology (IJERT) 2(8) 404-412.

[5] Rustum IM and Soliman HM 1988 Journal of Heat Transfer 110(2) 366-372.

[6] Sparrow EM and Bahrami PA 2004 International Journal of Heat and Mass Transfer 23(11) 1555–1560.

[7] Myhren JA and Holmberg S 2011 International Journal of Thermal Sciences 50(2) 115-123.

[8] Petroski J 2013 ASME 2013 International Technical Conference and Exhibition on Packaging and Integration of Photonic Microsystems 16-18 July Burlingame California USA.

[9] Nagarani N and Mayilsamy K 2010 International Journal of Engineering Science and Technology 2(7) 2839-2845.

[10] Yoshida M, Ishihara S, Murakami Y, Nakashima K and Yamamoto M 2006 JSME International Journal Series B- Fluids and Thermal Engineering 49(3) 869-875.

[11] Kundu B and Bhanja D 2010 International Journal of Heat and Mass Transfer 53(1-3) 254-267.

[12] Pise A and Umesh A 2010 International Journal of Mechanical Engineering and Technology (IJMET) 1 238-247.

[13] Kumbhar DG, Sane DNK and Chavan ST 2009 Proceedings of the International Conference on Advances in Mechanical Engineering 3-5 August Gujarat India.

[14] Babu G and Lavakumar M 2013 IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) 7(4) 24-29.

[15] Abu Hijleh AKB 2003 ASME Journal of Heat Transfer 125(5) 804-811.

[16] Matkar M and Ravanan PM 2011 International Conference on Operations and Quantitative Management 28-30 June Nasik India.

[17] Fernando Illan and Alarcon M 2010 Journal of Thermal Science and Engineering Applications 2(4) 041002 (7 pages).

[18] Sood Pakdee D, Behnia M and Copeland DW 2001 The International Journal of Microcircuits and Electronic Packaging 24(1) 77-83.

[19] Wange S and Metkar R 2013 International Journal of Engineering and Innovative Technology 2(11) 328-333.

[20] Gupta D and Wankhade SR 2015 International Journal on Mechanical Engineering and Robotics (IJMER) 3(2) 1-4.

[21] Sonkar A, Singh Rajput I, Dhruw J and Sahu KK 2017 International Journal of Engineering Research & Technology (IJERT) 6(4) 324-331.

[22] Sagar P, Teotia P, Sahlot A and Thakur HC 2017 Materials Today: Proceedings 4(8) 8558-8564.

[23] Jagdale M, Nitnaware P and Bharati R 2018 International Research Journal of Engineering and Technology (IRJET) 5(5) 2560-2562.