Maximizing The Heat Transfer Rate By Changing The Fin Geometry Using CFD As A Tool

Hardik S Rajput¹ Vivek B Patel²
¹Pg Student ²Assistant Professor
^{1,2} Mechanical Engineering Department, Lcit, Bhandu.

Abstract—Motorcycle engine releases heat to the atmosphere through the mode of forced convection heat transfer. To solve this, fins are provided on the outer part of the cylinder. The heat transfer rate is defined depending on the velocity of vehicle, fin geometry and the ambient temperature. Increasing the temperature difference between the object and the environment, increasing the convection heat transfer coefficient, or increasing the surface area of the object increases the heat transfer. Many experimental methods are available in literature to analyze the effect of these factors on the heat transfer rate. However, CFD analysis will be used to simulate the heat transfer of the engine block. ANSYS software is selected to run the simulation.

I. INTRODUCTION

As the fossil fuel reserves are depleting day by day, increasing of fuel price raising the technology towards new inventions and research, which provides engines which are highly efficient and produces high specific power. Air cooled engines are phased out and are replaced by water cooled engines which are more efficient, but almost all two wheelers uses Air cooled engines, because Air-cooled engines are only option due to some advantages like lighter weight and lesser space requirement. The heat generated during combustion in IC engine should be maintained at higher level to increase thermal efficiency, but to prevent the thermal damage some heat should remove from the engine.

Many studies, experiments are carried out on an IC Engine cylinder with fins using wind tunnel setup. The IC Engine is initially heated to 150°c and cooing rate of cylinder and fin is analyzed by varying the air velocity from 0 to 20 km/h using wind tunnel. This study is numerically extended for analysis of fin parameters using commercially available CFD code ANSYS Fluent. The numerically predicted results are validated with the experiments carried out in the laboratory. Hence the numerical study can also be extended to study the effect of fin pitch, fin thickness, normal and tapered fins, effect of holes and slits in fins etc.

II. MATERIAL AND METHOD

In the present studies, comparison is carried out on an IC engine cylinder including straight fins with cylinder including wavy fins. The IC engine is constantly heated to 400°c and cooing rate of cylinder and fin is analyzed by varying the air velocity from 20 to 60 km/hr. numerically for analysis of fin parameters using commercially available CFD code ANSYS. Hence the numerical study can also be extended to study the effect of fin pitch, fin thickness, normal and tapered fins, effect of holes and slits in fins.

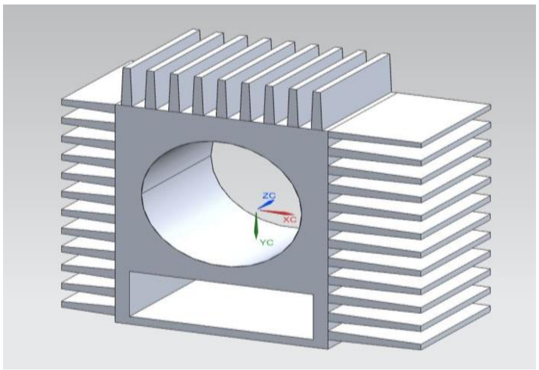

Fig. 1: .Four stroke SI engine cylinder with straight fins

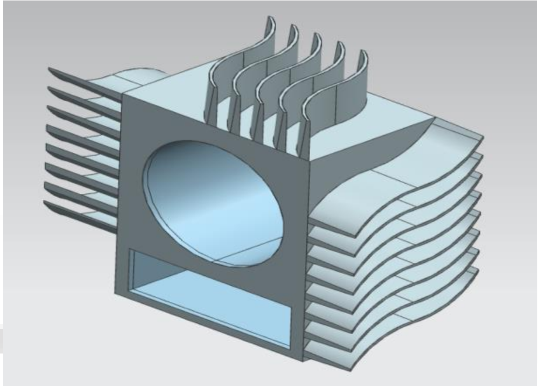

Fig. 2: .Four stroke SI engine cylinder with curve fins

A. Computational Modeling

Figure 1 and 2 shows the modeled cylinder with straight and curve fins which was used for CFD analysis. The dimensions of cylinder and fins are taken with the help of reverse engineering. A standard four stroke SI engine cylinder is measured with precise instruments. The bore diameter of the cylinder is Ø55mm. The cylinder with wavy fins is assumed with the help of standard engine dimensions.3-D Models are created in Pro-E wildfire 4.0. The output assembly design is created in parasolid format file for grid generation in ANSYS-ICEM CFD meshing tool.

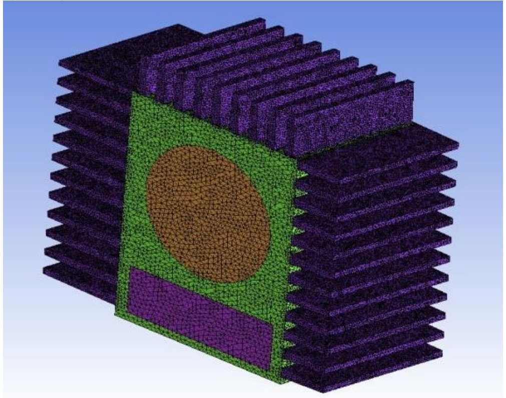

Fig. 3: Mesh of the Engine cylinder with straight fins

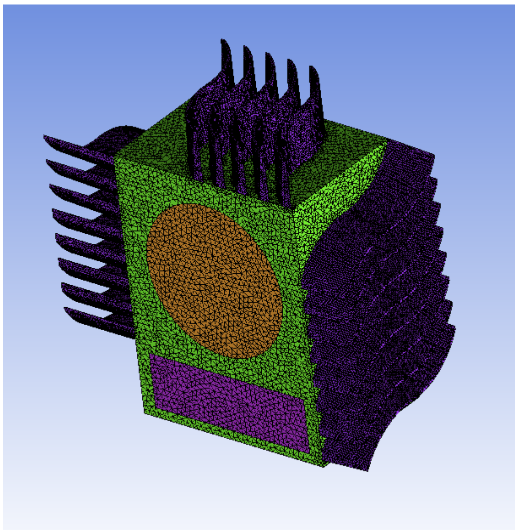

Fig. 4: Mesh of the Engine cylinder with curve fins

B. Grid Generation

The 3-D model is then discretized in ICEMCFD meshing tool. In order to capture both the thermal and velocity boundary layers the entire model is discretized using tetrahedral mesh elements which are accurate and involve less computation effort. Fine control on the tetrahedral mesh near the wall surface allows capturing the boundary layer gradient accurately. The entire geometry is divided into two domains AIR as. FLUID DOMAIN and ALLUMINIUM CYLINDER as SOLID DOMAIN. Once the meshes are checked for free of errors and minimum required quality. It is exported to ANSYS CFX pre-processor. Figure 3 and 4 shows the mesh of Engine cylinder. Figure 5 shows the Air domain.

The image part with relationship ID rId18 was not found in the file.

Fig.. 5: Air domain around the Engine cylinder

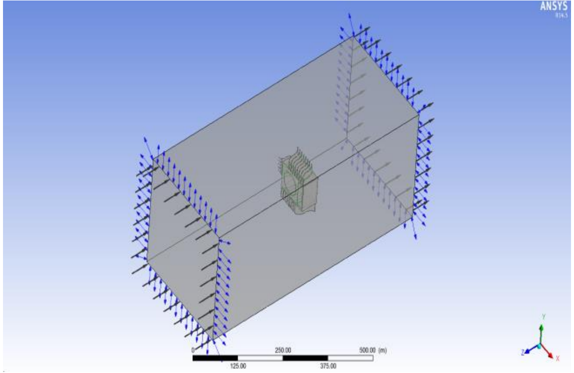

Fig. 6: .Boundary conditions to be taken on cylinder

C. Boundary Condition Setup

Air is assumed to be incompressible fluid. Ambient temperature and pressure are assumed as 298 K and 1.01325 Pa respectively. The value of the boundary conditions like operating temperature is 400 ^{0}C and Air velocity is taken 20 to 60 km/hr. Other boundary conditions like density, specific heat, thermal conductivity and other material properties are considered as constants throughout the analysis. The mesh is imported to ANSYS CFX and then the domains are initialized. The boundary conditions and the interface cylinder and air are set in the solver. Figure 6 shows the boundary conditions. The top and bottom of the cylinder surface are assumed to be adiabatic as it is insulated as per the experiment. Air domain is initialized at a temperature of 298 K as the initial temperature of the domain as per the experiment. The heat transfer takes place due to natural convection and conduction up to 298 K. The heat release from Cylinder with straight and wavy fins is calculated.

III. RESULTS AND DISCUSSIONS

The experimental results show that the value of heat release through cylinder, with straight and curve fins. at air velocity of 20 km/hr is 186.22K and 221.583K. and at air velocity of 40 km/hr is226.893K and 269.372K. At air velocity of 60 km/hr is 261.097K and 295.952K the heat release from the cylinder was obtained by multiplying the heat capacity of the heat storage liquid by the difference between 298°K and the temperature at outlet.

Heat release: Q = mcpΔT

Where, m = mass of the heat storage liquid, in Kg,

Cp = specific heat capacity of Air, in J/kg K,

ΔT = change in temperature at inlet and outlet, in K

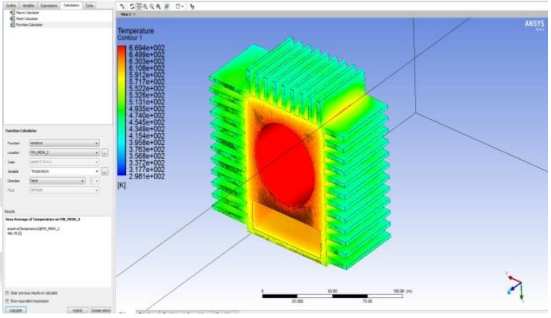

Fig. 7: Temperature contour of Engine cylinder straight fins at air velocity of 20 km/hr

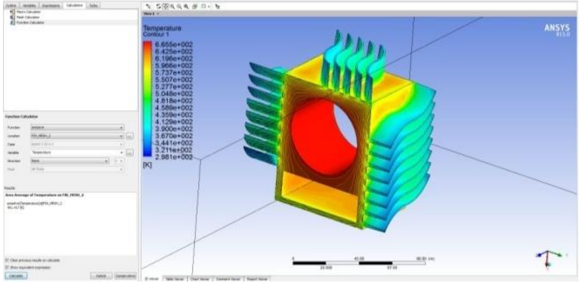

Fig. 8: Temperature contour of Engine cylinder curve fins at air velocity of 20 km/hr

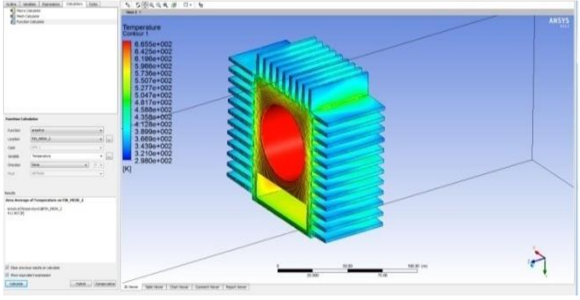

Fig. 9: Temperature contour of Engine cylinder straight fins at air velocity of 60 km/hr

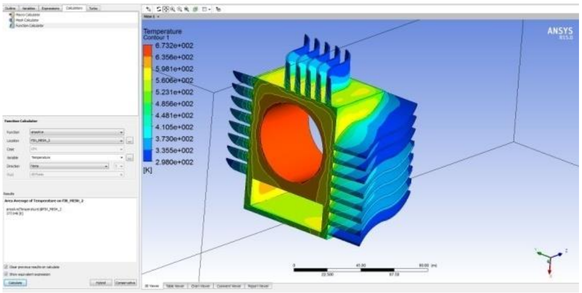

Fig. 10: Temperature contour of Engine cylinder curve fins at air velocity of 60 km/hr

It is seen that by changing cross-section and geometry of fin there is an increase in heat release. The temperature distribution profile of whole assembly in front view is shown in figure 7 to 10. The temperature of air, cylinder and fins which are at ambient temperature is also increased which conforms the heat transfer physics. Figure 11 shows the hear release from both the cylinder models at various air velocities. Up to Air velocity of 60 km/hr Heat release rate increases but after that it decreases because of higher velocity. The time of contact between the air and fins becomes less at higher velocities which leads to lower dissipation

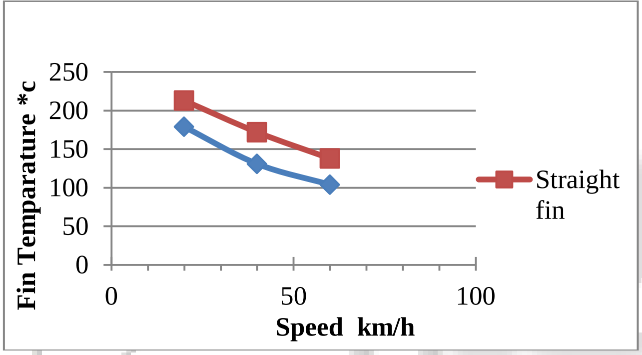

Fig. 11: Heat Release at Different Velocities for both the Models

IV. CONCLUSION

The summary of the present experimental work is as follows.

- The fin geometry and cross sectional area affects the heat transfer co efficient. In High speed vehicles curve fins provide better efficiency. Curve passage between the fins resulted in swirls being created which helped in increasing the heat transfer. Large number of fins with less thickness can be preferred in high speed vehicles than thick fins with less numbers as it helps inducing greater turbulence and hence higher heat transfer.
- Heat transfer coefficient can be increased by increasing the surrounding fluid velocity by forced convection. Heat transfer dependence on different stream velocities. But higher velocities also lead to lower heat transfer. So it is necessary to maintain fluid velocities around the fins.

ACKNOWLEDGMENT

We would like to sincerely acknowledge the uncourageous efforts of Mechanical Engineering Department LCIT bhandu. Our heartfelt thanks to faculty members who helped us in prepare research paper and give direction with their precious suggestions & rich experience.

REFERENCES

[1] Masao Yoshida, Soichi Ishihara,Yoshio Murakami, Kohei Nakashima,Masago Yamamoto,“Air cooling Effects of fins on a Motor cycle Engine”, JSME International Journal,Series B,Vol.49,No.3,2006

[2] Magarajan U., Thundil karuppa Raj R. and Elango T.“ Numerical Study on Heat Transfer of Internal Combustion Engine Cooling by Extended Fins Using CFD”, Research Journal of Recent Sciences ISSN 2277-2502 Vol. 1(6), 32-37, June (2012)

[3] J.Ajay Paul,Sagar Chavan Vijay, U.Magarajan & R.Thundil Karuppa Raj“Experimental and Parametric Study of Extended Fins In The Optimization of Internal Combustion Engine Cooling Using CFD”, International Journal of Applied Research in Mechanical Engineering (IJARME) ISSN: 2231 –5950, Volume-2, Issue-1, 2012

[4] Pulkit Agarwal, Mayur Shrikhande and P. Srinivasan,“Heat Transfer Simulation by CFD from Fins of an Air Cooled Motorcycle Engine under Varying Climatic Conditions”, Proceedings of the World Congress on Engineering 2011 Vol. III WCE 2011, July 6 - 8, 2011, London, U.K.

[5] Kumbhar D.G.· Dr. N K Sane,chavan S.T.,“Finite Element Analysis And Experimental Study Of Convective Heat Transfer Augmentation From Horizontal Rectangular Fin By Triangular Perforations”,Proc of the International conference on Advances in Mechanical Engineering, August 2009 S.V.N.I.T-SURAT

[6] .Nagarani, K.Mayilsamy ,“Experimental Heat Transfer Analysis On Annular Circular And N Elliptical Fins ” , International Journal of Engineering Science and Technology Vol. 2(7), 2010, 2839-2845

[7] Matkar M.V. ,P. M. Ravanan,“Thermal Analysis of Copper Fin by FEA”, ICOQM-10 June 28-30, 2011

[8] G.Raju, Dr. Bhramara Panitapu, S. C. V. Ramana Murty Naidu. International Journal of Modern Engineering Research (IJMER) www.ijmer.com Vol.2, Issue.6, Nov-Dec. 2012 pp-4516-4520 ISSN: 2249

[9] F. Illán, M. Alarcón.“Numerical analysis of combustion and transient heat transfer processes in a two-stroke SI engine” Applied Thermal Engineering 30 (2010) 2469e2475.

[10] S.C.Arora, S.Domkundwar ‘A Course in Heat & Mass Transfer’, Edition 7.07,2012, by Dhanpat rai & Co.

[11] R K Rajput ‘Heat & Mass Transfer’, ISBN-13, ISBN-10, 4th Edition,2001,Published by S. Chand Publisher

[12] C P Kodandaraman ‘Fundamentals of Heat & Mass Transfer’, ISBN-13, ISBN-10, 7th Edition,2010,Published by New Age International

[13] Fluent Incorporation, 2006. Fluent 6.3. User’s Guide. Canterra Resource Park 10 Cavendish Court Lebanon: Fluent Inc.