CFD analysis of engine cylinder fin with various materials

Arjun Vilay
arjunvilay@gmail.com

Rajiv Gandhi Proudyogiki Vishwavidyalaya, Bhopal, Madhya Pradesh

Vivek Tiwari
vtiwari3993@gmail.com

Rajiv Gandhi Proudyogiki Vishwavidyalaya, Bhopal, Madhya Pradesh

Prem Shankar
psdwivedi000@gmail.com

Rajiv Gandhi Proudyogiki Vishwavidyalaya, Bhopal, Madhya Pradesh

Sourabh Khambra
sourabhkhambra1994@gmail.com

Rajiv Gandhi Proudyogiki Vishwavidyalaya, Bhopal, Madhya Pradesh

ABSTRACT

The results of such paper of study and analysis, heat transfer and pressure loss of different shapes of the heat sink with the rectangular duct surface time zone are the same. There are two shaped fins are used by the analysis of the rectangle the shark fin, cylindrical (round). The purpose of the study was to determine the optimal size and shape of the rectangular longitudinal fins, Cylindrical Pin Fin, including horizontal thermal conductivity. This analysis has been completed the calculated maximum heat transfer rate of the fin surface and minimum pressure loss in the pipeline as a result of the shape of the change. The results of various calculations of different Nusselt no for Laminar and turbulent. After solving the problem of post-processing after the completion of the discovery of different results as outline figure, the X-Y plot and vector drawing the Laminar and turbulent flows including the heat transfer rate and pressure loss. In the light of the discussions on the basis of the results, print data concluded that heat transfer is the smallest rectangle shaped fins on the surface and the large round Pin Fin surface and pressure loss is the minimum piping, round the fins to enable it to make better use of the maximum heat transfer rate is required.

Keywords: Laminar flow, Turbulent flow, Nusselt number, Fins, Pressure loss

1. INTRODUCTION

In recent years there has been great demand for high-performance, lightweight, 'compact, and economical heat transfer components. The fins are recognized as one of the most effective means of increasing the heat dissipated. The design criteria of fins are different for various applications, but the primary concern is weight and cost. Therefore it is highly desirable to optimize the size of fins. The optimum dimensions are those for which maximum heat is dissipated for a given weight or mass of the fin.

The most effective heat transfer enhancement can be achieved by using fins as elements for heat transfer surface area extension. IC engine is a heat transfer fluid to occur in the engines themselves, usually the burning of fuel and air the oxygen content of the air. Internal combustion engines use thermal conversion of the energy of the fuel. In IC engine fuel energy into motive force. And after converting the heating power supply excess heat must be removed from the loop. Thermal will move to the atmosphere means that a fluid with water and air. The engine of the heat will move to the atmosphere of the fluid temperature is low. As a result of the combustion process. Engine temperature does not meet the entire power supply. If excess heats are never deleted, engine parts due to excessive temperature. Heat from the areas of high-temperature areas of low temperature as shown in the following figure the area below. When engine oil is oxidized (burning) heat generation. The additional heat generated by friction between moving parts. Only about 30% of the energy released is converted into useful work. The rest of the form (70 percent) must be removed from the engine to prevent parts melted. The main purpose is to project the analysis of thermal properties of different geometry as the square and round holes of the shark fin, materials and speed changes cylinder fin. Transient thermal analysis to determine if the temperature and other thermal quantity change over time. Different temperature distribution with the passage of time, the interest in many applications (such as in cooling). Accurate Thermal simulation to allow the key design parameters must be to improve their standard of living.

The fin performance has been obtained in an analytical form so that classical techniques can be adopted for optimization. In the present paper, a method has been suggested for optimizing longitudinal, radial (Circular) based on a CFD.
2. LITERATURE REVIEW

“Masao YOSHIDA, Soichi ISHIHARA, Kohei NAKASHIMA: Air-Cooling Effects of Fins on a Motorcycle Engine”[1] The author had developed the experimental cylinder for an air-cooled engine and the effects of the number of fins, fin pitch and wind velocities on cylinder cooling were investigated. The major results obtained are to increase the cylinder cooling; the cylinder should have a greater number of fins. But not more at a lower speed the airflow separated on the fin surface at the leeward side and the temperature on the fin surface increased there. The higher temperature on the local fin surface makes cylinder bore a greater deformation, as a result, scuffing and increased lubricating oil consumption may occur. [1]

“Pratima S. Patil, S.N. Belsare, Dr. S. L. Borse Analysis of internal combustion engine heat transfer rate to improve engine efficiency, specific power & combustion performance prediction.” This paper focuses on the substantial difference of heat flux exists for various places in the cylinder of an engine. A substantial difference of heat flux exists for various places in the cylinder of an engine. Maximum heat flux in each part occurs when the pressure in the cylinder is maximum. Heat flux on the intake valves is higher than another place of the cylinder. Heat flux on the cylinder head is more than a piston and it has the lowest value on the cylinder liner.

G.b Raju, Dr. Bharama Panitapu, S. C. V. Ramana Murty Naidu. “Optimal Design of an IC engine cylinder fin array using a binary coded genetic algorithm”.[3]This study also includes the effect of spacing between fins on various parameters like total surface area, heat transfer coefficient and total heat transfer. The aspect ratios of a single fin and their corresponding array of these two profiles were also determined. Finally, the heat transfer through both arrays was compared on their weight basis. Results show the advantage of triangular profile fin array.

Heat transfer through triangular fin array per unit mass is more than that of heat transfer through rectangular fin array. Therefore the triangular fins are preferred than the rectangular fins for automobiles, central processing units, airplanes, space vehicles etc. Where weight is the main criteria. At wider spacing, shorter fins are more preferred than longer fins. The aspect ratio for an optimized fin array is more than that of a single fin for both rectangular and triangular profiles.

Nagarajan U. , Thundil karrupa Raj R. , Elango T. “ Numerical study on heat transfer I C Engine cooling by extended fins using CFD” [4]In this study, heat release of an IC engine cylinder cooling fins with six numbers of fins having pitch of 10 mm and 20 mm are calculated numerically using commercially available CFD tool Ansys Fluent. The IC engine is initially at 150 and the heat release from the cylinder is analyzed at a wind velocity of 0 km/h.

It is observed from the CFD result that it takes 174.08 seconds (pitch=10mm) and 163.17 sec (pitch =20mm) for ethylene glycol domain to reach a temperature of 423 K to 393 K for initially. The experiment results show that the value of heat release by the ethylene glycol through cylinder fins of pitch 10mm and 20mm is about 28.5W and 33.90 W. [4]

Mr. N. Phani Raja Rao, Mr. T. Vishnu Vardhan. “Thermal Analysis of Engine Cylinder Fins By Varying Its Geometry and Material.”[5] The principle implemented in the project to increase the heat dissipation rate by using the invisible working fluid nothing but air. The main aim of the project is to varying geometry, material. In the present study, Aluminium alloy 6061 and magnesium alloy are used and compared with Aluminium Alloy A204.

The various parameters (i.e., shape and geometry of the fin) are considered in the study, shape (Rectangular and Circular), thickness (3 mm and 2.5 mm). By reducing the thickness and also by changing the shape of the fin to circular shaped, the weight of the fin body reduces thereby increasing the heat transfer rate and efficiency of the fin. The weight of the fin body is also reduced when Magnesium alloy is used. The results show, by using a circular fin with material Aluminium Alloy 6061 is better since heat transfer rate, Efficiency and Effectiveness of the fin are more. By using circular fins the weight of the fin body reduces compare to existing engine cylinder fin.

Heat Transfer Augmentation of Air Cooled 4 strokes SI Engine through Fins- A Review Pape. [8] The author had study number of the research paper and concludes that the phenomenon by which heat transfer takes place through engine fins must frequently be improved for these reasons.

N. Nagarani and K. Mayilsamy, Experimental heat transfer analysis on annular circular and elliptical fins.”[9]This other had analyzed the heat transfer rate and efficiency for circular and elliptical annular fins for different environmental conditions. Elliptical fin efficiency is more than a circular fin.

If space restriction is there along one particular direction while the perpendicular direction is relatively unrestricted elliptical fins could be a good choice. Normally heat transfer coefficient depends upon the space, time, flow conditions and fluid properties. If there are changes in environmental conditions, there is a change in heat transfer coefficient and efficiency also. [9]

3. METHODOLOGY

Computational fluid dynamics (CFD) is a computer-based simulation method for analyzing fluid flow, heat transfer, and related phenomena such as chemical reactions. This project uses CFD for analysis of flow and heat transfer. Some examples of application areas are aerodynamic lift and drag (i.e. airplanes or windmill wings), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses carried out in the various industries are used in R&D and manufacture of aircraft, combustion engines, as well as many other industrial products.
It can be advantageous to use CFD over traditional experimental-based analyses, since experiments have a cost directly proportional to the number of configurations desired for testing, unlike with CFD, where large amounts of results can be produced at practically no added expense. In this way, parametric studies to optimize equipment are very inexpensive with CFD when compared to experiments.

This section briefly describes the general concepts and theory related to using CFD to analyze fluid flow and heat transfer, as relevant to this project. It begins with a review of the tools needed for carrying out the CFD analyses and the processes required, followed by a summary of the governing equations and turbulence models and finally a discussion of the discretization schemes and solution algorithms is presented.

To run a simulation, three main elements are needed:

1. Pre-processor: A pre-processor is used to define the geometry for the computational domain of interest and generate the mesh of control volumes (for calculations). Generally, finer the mesh in the areas of large changes, the more accurate the solution. The fineness of the grid also determines the computer hardware and calculation time needed. The open-source pre-processor used for this project is called Salomé.

2. Solver: The solver makes the calculations using a numerical solution technique, which can use finite difference, finite element, or spectral methods. Most CFD codes use finite volumes, which is a special finite difference method. First, the fluid flow equations are integrated over the control volumes (resulting in the exact conservation of relevant properties for each finite volume), then these integral equations are discretized (producing algebraic equations through converting of the integral fluid flow equations), and finally an iterative method is used to solve the algebraic equations. (The finite volume method and discretization techniques are described more in the next sections. Open FOAM CFD code is used for solving the simulations in this project).

3. Post-Processor: The post-processor provides for visualization of the results, and includes the capability to display the geometry/mesh, create a vector, contour, and 2D and 3D surface plots. Particles can be tracked throughout a simulation, and the model can be manipulated (i.e. scaled by changing, rotating, etc.), and all in full color animated graphics. ParaView is the open-source post-processor used for this project.

The steps in the SIMPLE algorithm, are summarized as:

START
Estimate a starting guess for the pressure field p*.

STEP 1
Solve the discretized momentum equations for the velocity components, using the line- TDMA method, and based on the pressure guess p*. For a three-dimensional case as the one in this project, 6 discretized momentum balances are solved for each of the six neighbors for node P (W, E, S, N, B, and T). This step results in finding values for u*, v*, w*, based on p*.

STEP 2
Solve for pressure correction equation to find the pressure correction p". This is done with the discretized continuity equation by finding the mass imbalance bP and between total mass flow inflow and the total mass outflow of the six „guessed“ velocities (calculated based on the guessed pressure p*).

STEP 3
Correct the pressure and velocity components using the pressure correction, where the correction (p") is added to the initial guess (p*), to get the new pressure field p, or:

Equation 1 p = p* + p'

This step of the SIMPLE algorithm results in calculated values for velocity components and pressure: p, u, v, w, φ*, after correcting the guesses, which satisfy the continuity equation.

STEP 4
Solve the other discretized transport equations using the line-TDMA method to get calculated values for the remaining scalar variables φ.

3.1 The PISO Algorithm
The PISO (Pressure Implicit with Splitting of Operators) algorithm can be considered a continuation of the SIMPLE algorithm. PISO also involves making a guess for pressure, p*, and calculating the velocity components u*, v*, and w* based on this p*. There is a correction step to pressure and velocity fields, just as with the SIMPLE method. However, a second, additional, corrector step using the results from the first correction is carried out just after calculating the first correction. When the values for corrected pressure and velocity fields (now corrected twice) are determined, the other scalar transport equations are solved. The PISO algorithm requires more storage for the extra correction step, resulting in more computational resources needed.

4. PROBLEM FORMATION

Engine type- air cooled 4 strokes single cylinder engine
Displacement- 97.2cc
Bore stroke- 50 ........ 49.5mm
Height between the top and bottom end face- 69mm
Compression ratio- 9.1 : 1
The diameter of the sleeve- 54.4 mm
Material- Al alloy 7075 and cast iron
Property of material- cast iron
Thermal Conductivity – 0.05 w/mmK
Specific Heat – 500 J/kg °C

© 2018, www.IJARIIT.com All Rights Reserved
Density – 7.1 g/cc
Property of material Al alloys 7075
Thermal Conductivity – 130 w/mk
Specific Heat – 0.963 J/g °C
Density – 2.8 g/cc
Circular perforated diameter - 1.1256mm
Square perforated diameter - 1mm
No of hole - 10 (square and perforated)

5. MODELLING AND DESIGN
The Design of different geometrical shape of Fins was in CATIA and Analysis did by the ANSYS FLUENT software. The computational domain consists of a rectangular volume of large dimensions containing the finned body at its Centre. It was focused on the fins and appropriate boundary conditions were applied at the domain ends to maintain continuity. A fine mesh has been created near the fins to resolve the thermal boundary layer which is surrounded by a coarse external mesh for better results and fast solution. A face mesh has been done by Tetrahedron element as shown in Figures:

![Fig. 1: Splendor fin existing model meshing](image1)
![Fig. 2: Engine fin as circular perforated meshing](image2)
![Fig. 3: Engine fin as square perforated meshing](image3)

6. PROBLEM SETUP INFLUENT
The flow around the fin has been solved at different airflow velocities and air temperatures are at 303.15 K. A three-dimensional steady-state heat transfer analysis has been done by assuming a constant temperature on the inner surface of the wall. For obtaining the relation between heat transfer coefficient and velocity, the temperature was maintained constant and the simulations were carried out. Heat transfer characteristics for different configuration of the pin were obtained.

Fluid flow - air
Material property - Al alloys 7075 and cast iron

INLET:
Velocity= 20km/hr
Velocity= 60km/hr
Initial temperature= 303.15k
Cylinder temperature- 503.15k
Initial pressure- 101325 Pa
Turbulence intensity- 1%
Hydraulic diameter- 24mm

OUTLET:
Gauge pressure= -85Pa
Turbulence intensity- 1%
Hydraulic diameter-11.5mm

Table 1: No. of nodes and element for material Al alloys and cast iron for existing model

<table>
<thead>
<tr>
<th>Meshing parameter</th>
<th>Velocity</th>
<th>Velocity</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>20km/hr</td>
<td>40km/hr</td>
<td>60km/hr</td>
</tr>
<tr>
<td>No of node</td>
<td>44850</td>
<td>44850</td>
<td>44850</td>
</tr>
<tr>
<td>No of element</td>
<td>143464</td>
<td>143464</td>
<td>143464</td>
</tr>
</tbody>
</table>
Table 2: No. of nodes and element for material Al alloys and cast iron for square and circular perforated fin model respectively

<table>
<thead>
<tr>
<th>Meshing parameter</th>
<th>Velocity 20km/hr</th>
<th>Velocity 40km/hr</th>
<th>Velocity 60km/hr</th>
</tr>
</thead>
<tbody>
<tr>
<td>No of node</td>
<td>45654</td>
<td>45654</td>
<td>45654</td>
</tr>
<tr>
<td>No of element</td>
<td>195292</td>
<td>195292</td>
<td>195292</td>
</tr>
<tr>
<td>Meshing parameter</td>
<td>Velocity</td>
<td>Velocity</td>
<td>Velocity</td>
</tr>
<tr>
<td>No of node</td>
<td>20km/hr</td>
<td>40km/hr</td>
<td>60km/hr</td>
</tr>
<tr>
<td>No of element</td>
<td>77388</td>
<td>77388</td>
<td>77388</td>
</tr>
</tbody>
</table>

7. RESULTS AND DISCUSSION
In this work comparative study of the different fin at different velocity have been analyzed. A commercial finite volume analysis package, ANSYS FLUENT 15.0 selected to perform numerical analysis on the model. The realizable green-gauss cell-based turbulence model with standard wall function was set for each model. The Segregated 3D solver with an implicit formulation was set to solve the models.

7.1 Temperature contour
Splendor fin existing Al alloys 7075 material
Velocity 20km/hr-pressure-temperature respectively

7.2 Splendor fin existing CI material:

Fig. 4: Contours of static temperature at a different velocity
7.3 Spender fin square perforated for CI material

7.4 Spender existing model

<table>
<thead>
<tr>
<th>S. NO.</th>
<th>Velocity (km/hr)</th>
<th>Temperature Of air in k</th>
<th>Pressure drop in Pa</th>
<th>Air heat transfer (q) in W</th>
<th>Convection co efficient h</th>
<th>Nu no</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>303.15</td>
<td>22.8021</td>
<td>1030.30</td>
<td>396.41</td>
<td>2268.7</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td>303.15</td>
<td>90.6275</td>
<td>1718.92</td>
<td>661.36</td>
<td>3785.1</td>
</tr>
<tr>
<td>3</td>
<td>60</td>
<td>303.15</td>
<td>203.007</td>
<td>2304.10</td>
<td>886.50</td>
<td>5073.6</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>S. NO.</th>
<th>Velocity (km/hr)</th>
<th>Temperature Of air in k</th>
<th>Pressure drop in Pa</th>
<th>Air heat transfer (q) in W</th>
<th>Convection coefficient h</th>
<th>Nu no</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>303.15</td>
<td>22.8021</td>
<td>1102.61</td>
<td>424.232</td>
<td>2472.9</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td>303.15</td>
<td>90.6275</td>
<td>1895.80</td>
<td>729.415</td>
<td>4174.5</td>
</tr>
<tr>
<td>3</td>
<td>60</td>
<td>303.15</td>
<td>203.007</td>
<td>2591.55</td>
<td>997.106</td>
<td>5706.6</td>
</tr>
</tbody>
</table>

Table 1: For cast iron material

Table 2: For AL alloy 7075 material
7.5 Splender square perforated fin

Table 3: Cast iron square cross section 1mm mm

<table>
<thead>
<tr>
<th>S. NO.</th>
<th>Velocity (km/hr)</th>
<th>Temperature Of air in k</th>
<th>Pressure drop in Pa</th>
<th>Air heat transfer (q) in W</th>
<th>Convection co efficient h</th>
<th>Nu no</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>303.15</td>
<td>22.6689</td>
<td>1063.52</td>
<td>409.193</td>
<td>2341.9</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td>303.15</td>
<td>90.5877</td>
<td>1768.74</td>
<td>680.528</td>
<td>3894.8</td>
</tr>
<tr>
<td>3</td>
<td>60</td>
<td>303.15</td>
<td>202.33</td>
<td>2365.30</td>
<td>910.054</td>
<td>5208.4</td>
</tr>
</tbody>
</table>

Table 4: Al alloys 7075 square cross section 1mm mm

<table>
<thead>
<tr>
<th>S. NO.</th>
<th>Velocity (km/hr)</th>
<th>Temperature Of air in k</th>
<th>Pressure drop in Pa</th>
<th>Air heat transfer (q) in W</th>
<th>Convection co efficient h</th>
<th>Nu no</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>303.15</td>
<td>22.6689</td>
<td>1124.28</td>
<td>432.56</td>
<td>2475.7</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td>303.15</td>
<td>90.5877</td>
<td>1925.64</td>
<td>740.894</td>
<td>4240.2</td>
</tr>
<tr>
<td>3</td>
<td>60</td>
<td>303.15</td>
<td>202.33</td>
<td>2624.95</td>
<td>1009.96</td>
<td>5780.1</td>
</tr>
</tbody>
</table>

7.6 Splender circuler perforated fin

Table 5: For cast iron material

<table>
<thead>
<tr>
<th>S. NO.</th>
<th>Velocity (km/hr)</th>
<th>Temperature Of air in k</th>
<th>Pressure drop in Pa</th>
<th>Air heat transfer (q) in W</th>
<th>Convection co efficient h</th>
<th>Nu no</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>303.15</td>
<td>21.6271</td>
<td>1037.79</td>
<td>399.295</td>
<td>2285.2</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td>303.15</td>
<td>86.005</td>
<td>1734.024</td>
<td>667.938</td>
<td>3822.7</td>
</tr>
<tr>
<td>3</td>
<td>60</td>
<td>303.15</td>
<td>192.626</td>
<td>2321.50</td>
<td>893.204</td>
<td>5111.9</td>
</tr>
</tbody>
</table>

Table 6: Al alloys 7075 cross-section 1mm mm

<table>
<thead>
<tr>
<th>S. NO.</th>
<th>Velocity (km/hr)</th>
<th>Temperature Of air in k</th>
<th>Pressure drop in Pa</th>
<th>Air heat transfer (q) in W</th>
<th>Convection co efficient h</th>
<th>Nu no</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>303.15</td>
<td>21.6271</td>
<td>1095.03</td>
<td>421.31</td>
<td>2411.3</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td>303.15</td>
<td>86.005</td>
<td>1880.91</td>
<td>723.68</td>
<td>4141.8</td>
</tr>
<tr>
<td>3</td>
<td>60</td>
<td>303.15</td>
<td>192.626</td>
<td>2565.01</td>
<td>986.89</td>
<td>5648.1</td>
</tr>
</tbody>
</table>

Fig. 7: Heat transfer vs. velocity

Fig. 8: Nusselt no vs. velocity

8. CONCLUSION
As we found a result, a cylinder fins SPLENDER institutions 97.2 cc motorcycle model using Parametric Software ANSYS 14.5 fluent the original model is accomplished by changing the geometry of the heat sink, the cross-section of the fins modifications as square and round hole, 10-hole in the middle of the length of the fins. The materials used for the current fin body is cast iron, but the materials used for the cast iron and AL alloy 7075. The original model of material changes through the consideration of their density and thermal conductivity. Low-density aluminum alloy 7075 compared to other cast material weight fin institutions rarely used 7075 aluminum alloy. The results of the analysis by observing the hot, heat transfer rate the square holes fin AL alloy 7075 speed 60 km/h is 2624.955 watts. This shows the effective results as compared to another existing model.
9. REFERENCES


