International Journal of Innovative Research in Engineering
Volume 4, Issue 1 (January-February 2023), PP: 195-206.
www.theijire.com **ISSN No: 2582-8746**

CFD Simulation of Heat Transfer In Air-Cooled Finned Four Stroke Engines

Amir Salim[1], Vijaykant Pandey[2]
[1] M-Tech Research Scholar, RKDF College of Technology, Bhopal, M.P, India.
[2] Assistant Professor, RKDF College of Technology, Bhopal, M.P, India.

How to cite this paper:
Amir Salim[1], Vijaykant Pandey[2], "CFD Simulation of Heat Transfer In Air-Cooled Finned Four Stroke Engines", IJIRE-V4I01-195-206.

Abstract: *The purpose of this experimental work is to study the forced conjugate heat transfer process taking place in engine cylinder block with varying wind speeds ranging from 40 km/hr. to 80km/hr., varying environmental conditions, various fin pitch and fin width. ADC12 Material is used by the Engine developers of 4-stroke. ADC12 aluminium alloy haschemicalcompositionofAl-Cu-Si-Mg.ThedensityofADC12alloyisverylesscomparedtocastironresulting less weight of the engine. Wehavedesign3Dmodel oftheactualenginecylinderblockof4-strokebikeEnginehasbeendrawn in Ansys 15.0. The Average heat transfer coefficient for the surface of the engine cylinder block has been calculated with the help of commercially available CFD tool ANSYS Fluent. Result shows that at 80 km/hr wind velocity heat transfer rate (120w/m^{2}K) in rectangular fin greater than Masao Yasida (118w/m^{2}K) in cylindrical fin.*

Key Word: *Forced convection, Engine cooling, Fin geometry, Wind Speed*

I.INTRODUCTION

In internal combustion engine the combustion of fuel (normally a petroleum products) take place with an oxidizer (usually environmentally air) in a combustion chamber. In an I.C. engine, theexpansionofthehightemperatureandhigh-pressuregasesgeneratedbycombustionwhichappliesdirect force to various component of the engine, i.e. pistons. This force moves the component i.epistonovera distance, generatedmechanicalenergyintheformofrotation.Aircooledenginesinheavy vehicles and cars are phased out and are replaced by water cooled engines which are more efficient, but almost all two wheelers uses Air cooled engines, becauseAir-cooledenginesareonlyoptionduetosomeadvantageslikelighterweight,lesser space requirement and lesser maintenance. The heat generation from combustion in IC engine should be maintained at higher level to increase thermal efficiency, but to prevent the thermal damage to the engine geometry some heat should be removed from the engine.

Fins are one of the heat transferring devices which are used to increase the heat transfer on engine cylinder walls. Therefore, it is necessary to study the heat transfer rate of the fins. The research has been done to increase fin efficiency by Changing fin material and fin geometry. The fins are generally extended surfaces of projections of materials on the Cylinder block and head. The fins are usually used to increase the heat transfer from the Cylinder to the surroundings air by increasing the heat transfer surface area.

The heat transfer rate depends upon the velocity of the vehicle and fin geometry and temperature of cylinder. The heat transfer also depends on air velocity and environmental temperature. If the surrounding temperature is low and wind velocity is high, it can result in over cooling of the engine, which can overcome the thermal efficiency of the engine. Whereas, If the surrounding temperature is high and wind velocity is low it can result in over heating of the engine, which can cause distortion of the engine geometry and break down of the engine functioning. Various experimental methods are available in literature to study the effect of that factors on the heat transfer rate.

There are many methods available in industry to design the engine cooling system. Few of them are experimental, analytical and numerical methods. The experimental heat transfer study involves constructing various cylinder-fingeometryandconductthestudyinwindtunnel.Butthatprocessis time consuming and costly. In today's rapidly moving and competitive world, the industry doesn't have time to invest lot of time and money to conduct experiment. There is a powerful alternative way called CFD tools, which is used to study and design the cooling system in engine efficiently. There are numerous CFD package available in market, to simulate the heat transfer phenomenon in engine fin assembly. In present study Ansys Workbench 15.0 (Fluent) is used to conduct numerical study. The reliability of the methodology proposed in this study has been validated with the experimental results. First the parametric 3D model of the simple cylinder with circular fins is constructed using CREO.

FINS

A fin is a surface that extends from an object to increase the rate of heat transfer to or from the environment by increasing convection. The amount of conduction, convection, radiation of an object determines the amount of heat it transfers. Increasing the temperature difference between the object and the environment, increasing the convection heat transfer coefficient, or increasingthesurfaceareaoftheobjectincreasestheheattransfer.Sometimesitisnoteconomicaloritisnotfeasible to change the first two options. Adding a fin to the object, however, increases the surface area and can sometimes be economical solution to heat transfer problems. Circumferential fins around the cylinder of a motor cycle engine and fins attached to condenser tubes of a refrigerator area few familiar examples.

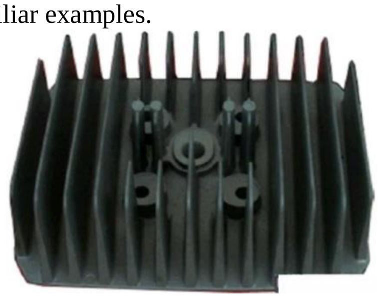

Figure1:Automobile/4strokeFin

Fin materials

- Cast iron
- Aluminumalloy
- Alusil
- Duralumin
- Comparison of alusiland duralumin

Types of fins

A fin is a surface that extends from an object to increase the rate of heat transfer to or from the environment by increasing convection. The amount of conduction, convection, or radiation of an object determines the amount of heat it transfers. Increasing the temperature difference between the object and the environment, increasing the convection heat transfer coefficient, or increasingthesurfaceareaoftheobjectincreasestheheattransfer.Sometimesitisnoteconomicaloritisnotfeasible to change the first two options. Adding a fin to an object, however, increases the surface area and can sometimes be an economical solution to heat transfer problems.

II.LITERATURE REVIEW

1. **Raman Kumar et. al.** A motorcycle engine cylinder's combustion chamber is subjected to high temperatures and thermal stresses, thus fins are added to cool the cylinder and fins are given on the cylinder to boost the heat transfer rate. A thermal examination of a finned engine block was undertaken in this paper. The heat dissipation inside the cylinder may be determined by doing a thermal study on the fins of the cylinder block. The idea behind cylinder block cooling is to enhance the heat transfer rate by lengthening the fins on the cylinder block. This paper balances the body materials down to the inner balance centre of amalgam and dark cast iron utilising the 100cc Platina engine head model and the Solidworks 3D screen design framework software programme to construct a set of engine head housing geometry was put together. At 300°C and 500°C, we utilised rectangular aluminium 6065.

2. **Sujan Shresthaet. Al.** The shape of the heating movement of internal combustion engines can be shown by different techniques. These techniques range from original worm systems to multidimensional differential condition representation. Blades are installed on the outside of the chamber to increase heat retention due to convection. Thermal studies of engine compartment blades are increasingly important to understand the heat dissipated in the compartment. An authored study shows that heat transfer improves with a wider surface and the heat transfer coefficient is affected by changes in blade cross-section. This research helps identify a better balance of geometry and materials for greater heat dissipation and engine cooling. We now use common materials like dark cast iron for engine square.

3. **NamanSahuet. al** The engine housing is one of the essential components of the engine and is exposed to various high and high temperatures. Blades are installed on the outside of the chamber to increase heat retention due to convection. Thermal inspection of motor housing blades is increasingly useful for understanding the heat dissipated within the housing. The current survey was conducted to improve information on various recent surveys. This shows that blade heat transfer depends on balance composition, balance pitch, balance design, wind speed, texture and atmospheric environment. Written experiments have shown that the heat transfer is improved by the extended surface and the heat transfer coefficient is affected by changes in the equilibrium cross-section. This research helps identify geometries and materials that balance higher heat dissipation and engine cooling.

4. **Charanet. al.** We have broken down a broad surface that is commonly used to promote convective heat transfer in a wide variety of design applications. The holes in the parallel sides of the blades are suitable for improving the speed of heat transfer. As a result of the investigation, it was found that aluminum materials with three triangular holes had the lowest tip temperature and aluminum materials with three triangular holes had the highest heat transfer. As a result of the inquiry, it

was found that the Nusselt number in the clamped scale increases when the blade is jammed in front of the blade that is jammed. Therefore, 3 aluminum triangles with horizontal holes are generally considered suitable for balance applications.

5. **K.Rama Chandra Manohar et al** The engine (SPLENDOR 150 CC) is one of the most important mechanical assemblies in a vehicle that is exposed to aircraft temperature and thermal instability. The balance, which changes as the operator cools down, is the basic expansion we're used to blowing expansion from our engines. The blades are used to reach the adjusted total from the plan to the environment. OPERATOR (SPLENDOR 150 CC) By achieving a calculated wind test on a cooled balance, currently in the Specialist Brilliance 150ccso, the balance is adjusted by inserting a modified type of tooth and is the aforementioned material. The capacity and balance capabilities are invaluable for competent Blade programming. The main point of our test is to activate the care breeze with the existing indentation and the test is completed by the ANSYS program.

6. **Beldaret. al.** Conducted continuous research with the help of CFD programming. Wind flow test and pressure drop test were done. The size of the points varies at 10%, 20%, and 30%, and the warm input varies from 25, 45, and 65 watts. In the uncompensated blade cluster region, the equilibrium surface is reduced, but the heat transfer enhancement is still reduced. The balance of payment cluster shows the focal substance of the balance again in the air of the virus, and the warm motion is increasing. Placing an indentation on the focal bit of the balance, after making a change to the conventional air flow example, significantly increases the peripheral velocity and changes the pneumatic stress across the channel, creating a tubular worm. The air temperature inside the sink channel will increase.

III.CFD

Computer primarily based simulation is mentioned during this chapter. Procedure simulation is technique for examining fluid flow, heat transfer and connected phenomena like chemical reactions. This project uses CFD for analysis of flow and warmth transfer. CFD analysis accepted go in the various industries is employed in R&D and producing of craft, combustion engines and in powerhouse combustion similarly as in several industrial applications.

Why computational simulation

Three-dimensional (3D) numerical analysis of whorled coil tubes is dispensed by victimization business CFD tool ANSYS 18.2. this can become troublesome and time overwhelming, if this analysis is dispensed by experimentation. Experimental setup is extremely expensive that's why in my work I take facilitate of CFD to create it easier and fewer time overwhelming

Computational fluid dynamics

Computational fluid dynamics, because the name implies, could be a subject that deals with procedure approach to fluid dynamics by means that of a numerical resolution of the equations that cause the fluid flow and though it's known as procedure fluid dynamics; it doesn't simply wear down the equations of the fluid flow, it's conjointly generic enough to be ready to solve at the same time along the equations that direct the energy transfer and similarly the equations that verify the chemical process rates and the way the chemical process takings and mass transfer takes place; of these things may be tackled along in a regular format. So, this define permits America to wear down a really complicated flow circumstances in fairly quick time, specified for a specific set of conditions, associate degree engineer would be ready to simulate and see however the flow is happening and what quite temperature distribution there's and what quite product area unit created and wherever they're fashioned, in order that {we can |we will |we area unit able to} build changes to the parameters that area unit below his management to switch the approach that these items are happening. So, therein sense procedure fluid dynamics or CFD becomes a good tool for a designer for associate degree engineer. it's conjointly a good tool for associate degree associate degree alysis for associate degree examination of a reactor or an instrumentality that isn't functioning well as a result of in typical industrial applications, several things is also happening associate degreed what a designer has had in mind at the time of fabricating or coming up with the instrumentality won't be really what an operator of the instrumentality introduces into the instrumentality at the time of operation, perhaps once 5 years or 10 years changes might need taken place in between; and in such a case, the presentation of the instrumentality won't be up to the quality and you'd wish to modify it in such some way that you just will restore performance. So, the question is then, what this can managed to the autumn within the performance associate degreed what quite measures we are able to build while not creating an overall adjustment within the finish of apparatus. Is it potential to urge improved performance from the equipment? Is it potential to extend the productivity? If you wish to appear on of these analysis, then procedure fluid dynamics is employed.

IV.METHODOLOGY

Steps to simulate the experiment

- Model the geometry proposed
- Import the step file of the above geometry in Ansys Design modeler.
- Create Air Domain around the cylinder fin geometry.
- Mesh cylinder fin and Air domain as single part. More mesh refinement is given for cylinder-fin and fluid around the

Surface. Successive coarser mesh is chosen away from the geometry for faster convergence.

- Apply boundary conditions and run simulation in Ansys Fluent.

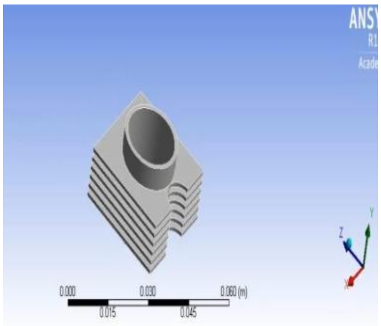

Figure 2:3D proposed model cylinder Geometry.

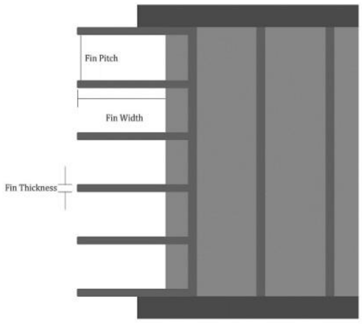

Figure3:Enginefinsterminology.

Specification of engine

The cylinder block of two-wheelerwith4-strokeengineis selected for analysis. The modeling of cylinder is carried out using Ansys software.

Sr.No.	Parameter	Value	Unit
1.	Displacement	124.7	cc
2.	Max Power	6.72	kW@ 7000 rpm
3.	Max torque	10.35	kW@ 4000 rpm
4.	Bore× Stroke	52.4 × 57.8	mm
5.	Compression Ratio	9.1: 1	Unitless

Table1: Specification of the proposed engine

MESHING: Tetrahedron mesh is generated throughout the air domain and hexahedron mesh is generated for the engine block. The tetrahedron mesh is unstructured mesh and it is used for faster calculation of complex convection equations. Hexahedron mesh is structured mesh and are good for solving heat conduction problems. Hence it is hybrid mesh of tetrahedronandhexahedron. The mesh is conformal as the two different types of elements matchupa long the interface. The coupled boundary condition is applied where solid and fluid interfaces meet. There are mainly two approaches of solving wall heat transfer problems. When the wall thickness is considerable, themes his also to be generated throughout the wall and energy equation is solved for conduction also. But if the wall thickness is negligible, it is considered as thin wall where meshing is not required and energy equation is not solved for wall.

Hexahedron meshes are generally more uniform. Hexahedron mesh can fill the geometry more efficiently than any other mesh shapes. But hex mesh always takes more man time than any other mesh. Here hex mesh for solid geometry is generated with the help of slice command. With slice the geometry is cut in such a way that each face is having 3or 5 boundary curves.

Maximum skewness is 0.85.Theminimumskewnessis3e-7.90%oftheelementshaveorthogonalquality of more than 0.8. The minimum orthogonal quality is 0.2. Statistic of skewness and orthogonal quality shows good quality of meshing. Very fine mesh is generated near the fins and surrounding fluid domain to capture the thermal boundary layer phenomenon and successive coarser mesh is generated away from the cylinder block for faster results as shown in figure.

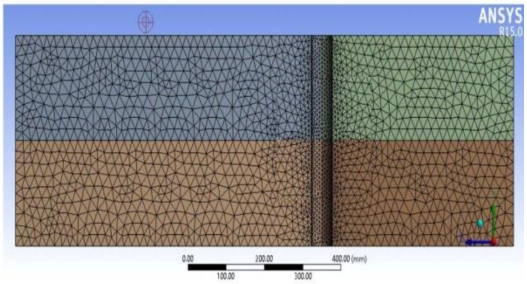

Figure-4 Mesh of cylinder block and Fluid Domain.

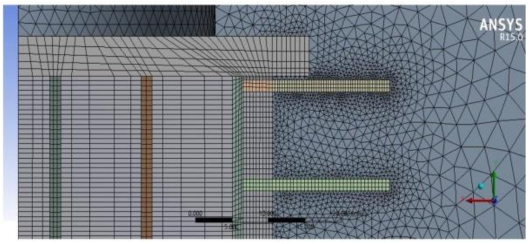

Figure5:Hexahedronmesh of cylinder block and tetrahedronmesh of fluid domain.

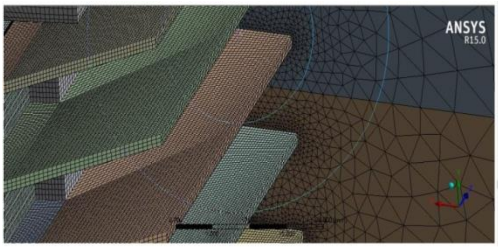

Figure 6: 3Dviewofthe cylinder air domain mesh interface.

Sr.No.	Boundary Condition	Parameter	Surface/Direction	Value
1	Inlet	Velocity,	Flows tream direction	40,50,60,70,80 km/hr
		Temperature	Air domain	300, 263,318 K
2	Outlet	Gauge pressure	outer face	0 pa
3	Wall	No slip	top, bottom, side and geometry surface	Adiabatic wall
4	Base Temperature	Temperature	Cylinder block inside wall	463K

Table2: Boundary Conditions for problem setup

Fluid property

Type of fluid-	Air
Density	1.225kg/m3 @288k
Viscosity	1.81x10^-5@ 288k
Specific heat-	Cv-.718 KJ/KgK Cp-1.00KJ/KgK
Thermal conductivity	K-1.4W/m°C

Table3:Fluid specification.

V RESULTS & DISCUSSION

The temperature contour clearly indicates that the temperature is low at stagnation region. The surface temperature is higher at the backward side of the engine. Observation of the temperature pattern can show the areas where higher thermal load happens. Special care while designing is to be taken for higher thermal load zones. Comparing the temperature pattern at surrounding temperature of27 ˚C and -10 ˚C, it is clear that at -10 ˚C, the higher heat loss is taking place. At windward side, the average surface temperature is lower compared to leeward side.

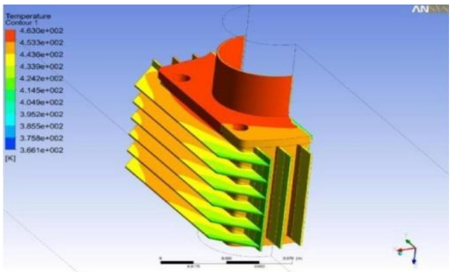

Figure7: Temperaturecontouratwindvelocityof60km/hrandsurroundingtemperatureof27 °C

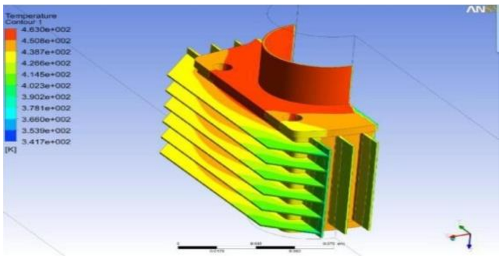

Figure 8:Temperature contourat windvelocityof60km/hrandsurroundingtemperatureof-10 °C

At smaller fin pitch the air flow is restricted to enter the space between two consecutive fins. It makes overlapping of the thermal boundary layers. This increases the temperature between the fins. Hence the heat released decreases. The total number of fins for 5mm and 6.5mm of fin spacing is 10. The interference of the fin boundary layer reduces the overall heat transfer coefficient. The temperature cont our for varying fin spacing i.e. fin pitch is as shown in figure. We can predict the temperature distribution over the fin surface. It is necessary to have uniform temperature distribution over the entire engine block. If there is drastic change in temperature of the geometry, then it can cause distortion in the engine. Hence special care is to be taken to reduce the thermal loads and thermal fatigue over the engine body.

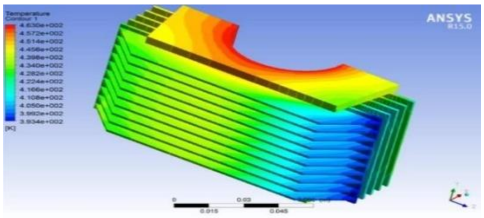

Figure9:Fin Spacingof5 mm, Temperature contour for various fin spacing.

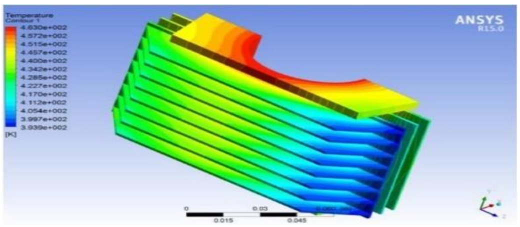

Figure10:Finspacingof8mm, Temperature contour for various fin spacing.

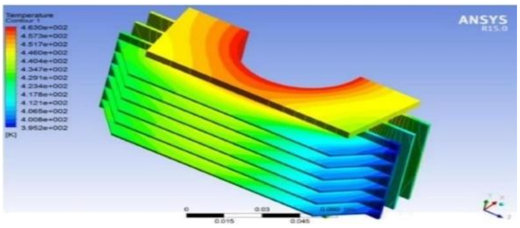

Figure 11:Finspacing of11mm,Temperature contour for various fin spacing

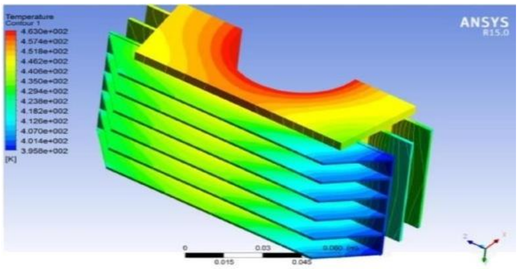

Figure12:Fin spacingof14mm,Temperature contourfor various fin spacing

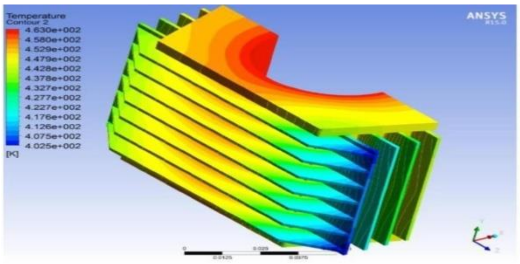

Figure 13:Fin Width= 5mm,TemperaturecontourforvariousFinWidth

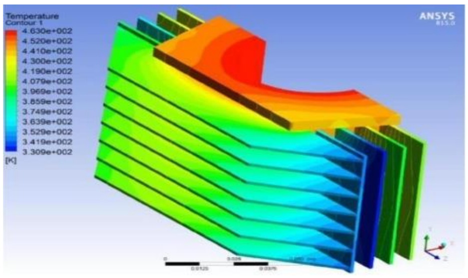

Figure14:Fin Width= 19mm,TemperaturecontourforvariousFinWidth

Wind flow over fins

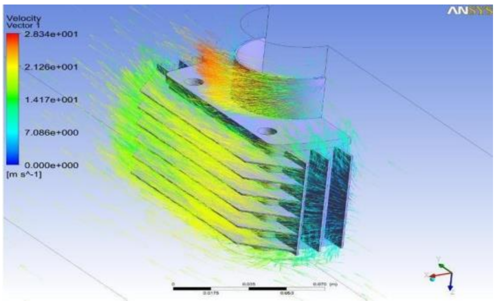

Figure15: Velocity Vectors at wind velocity of 60km/hr.

Different fin Pitch With velocity

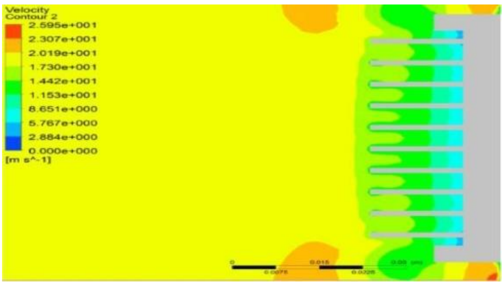

Figure16: Fin spacing of5mm

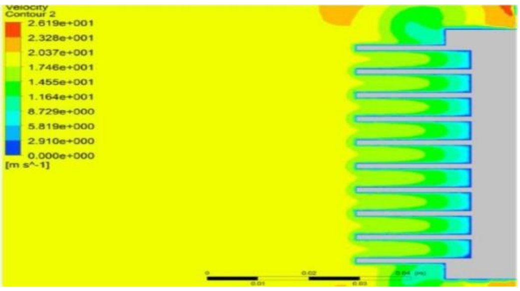

Figure 17: Fin Spacing of6.5 mm

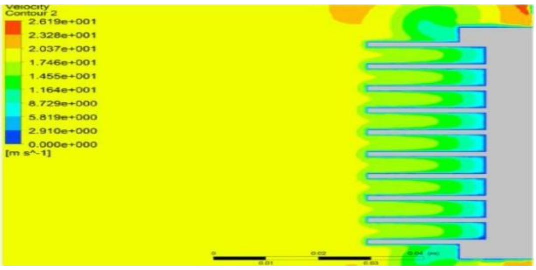

Figure 18: Fin spacing of 8 mm

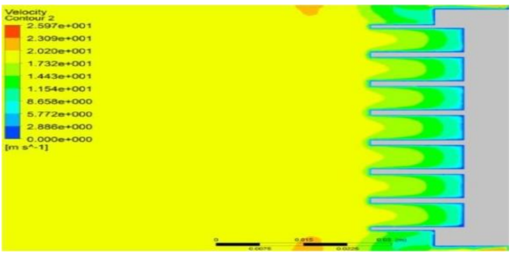

Figure19: Fin Spacing of9.5mm

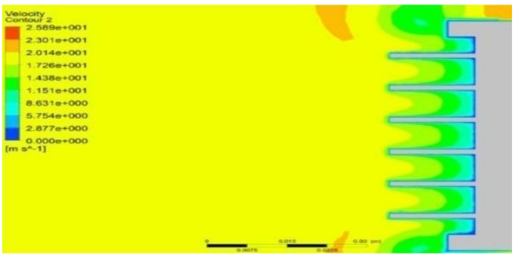

Figure20: Fin Spacing 11mm

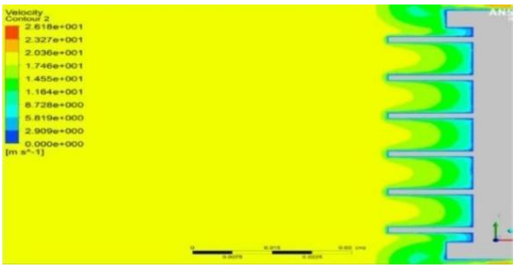

Figure21: Fig Spacing of12.5mm

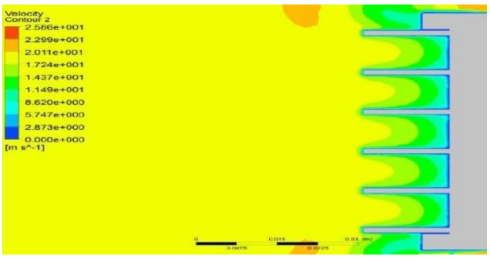

Figure22: Fin Spacing of14mm

Different fin Width with velocity

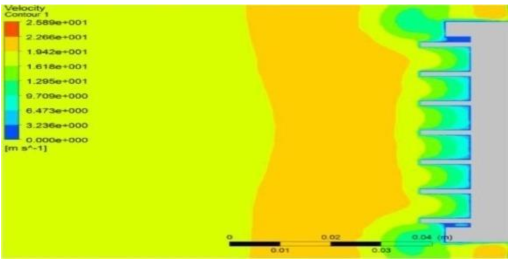

Figure23: Fin width = 5mm

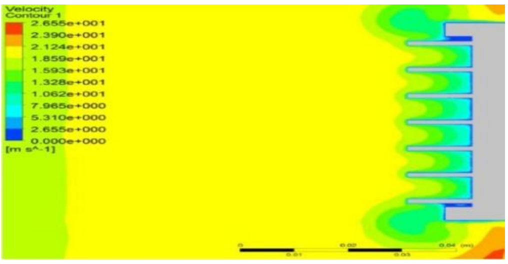

Figure24: Fin Width =7mm

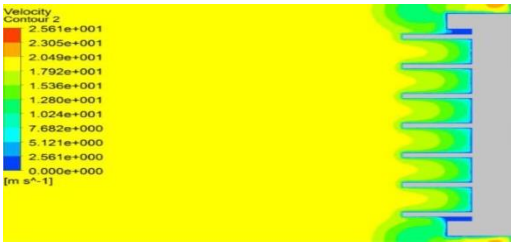

Figure25: Fin Width =9mm

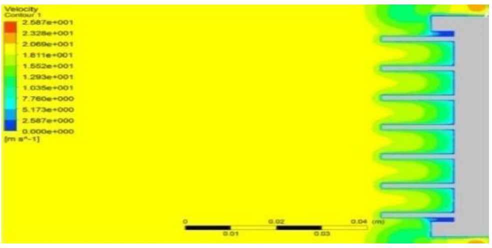

Figure26: Fin Width =11mm

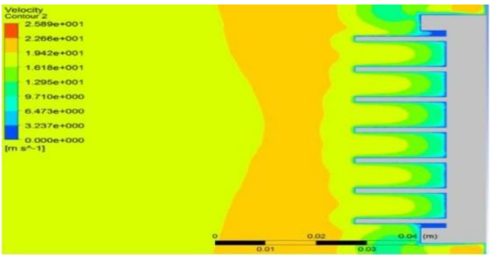

Figure27: Fin Width =13mm

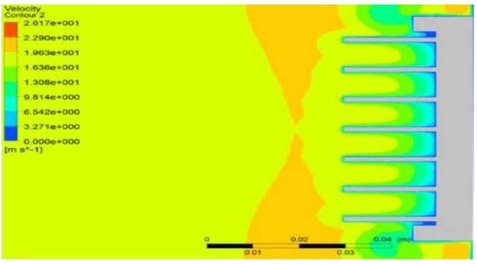

Figure28:Fin Width=15mm

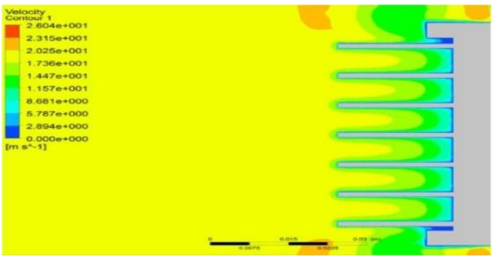

Figure29 Fin Width =17mm

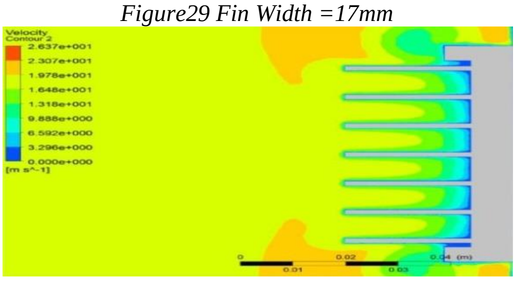

Figure30: Fin Width =19mm.

Comparative performance analysis

Plots of heat transfer parameters over wind velocity

The plot of surface heat transfer coefficient with varying wind velocities ranging from 40 km/hrto 80 km/hr. is as shown in figure. The surface Nusselt number value is higher ranging from2000to5500, which indicated forced convection phenol men on.

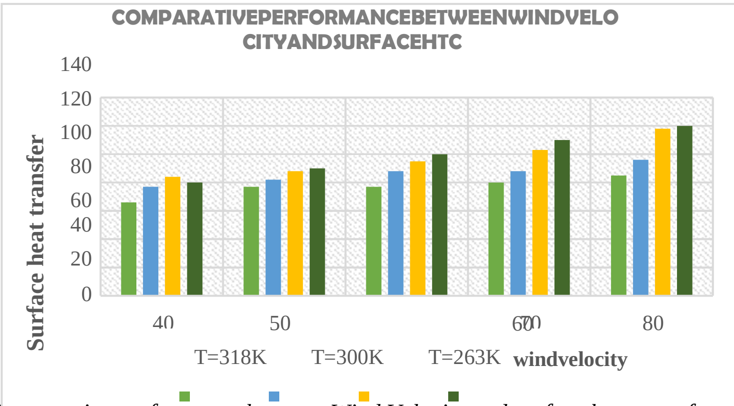

Figure 31: Comparative performance between Wind Velocity and surface heat transfer coefficient.

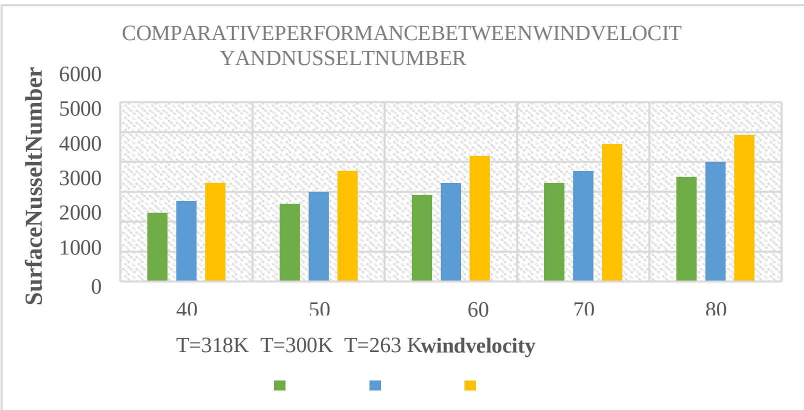

Figure 32:Comparative performance between Wind velocity vs Nusselt number

The empirical relation between the heat transfer coefficient and wind velocity is derived as below. The relation agrees with the relation given by pulkitet. el.:

$\alpha = 6.9 \times U^{0.597}$

In fluid Dynamics, Turbulent kinetic energy is the mean kinetic energy per unit mass associated with eddies in turbulent flow. The turbulent kinetic energy is characterized by measured root mean square velocity fluctuations. With increasing wind velocity, the turbulence eddies increase, which gives higher convective heat transfer coefficient.

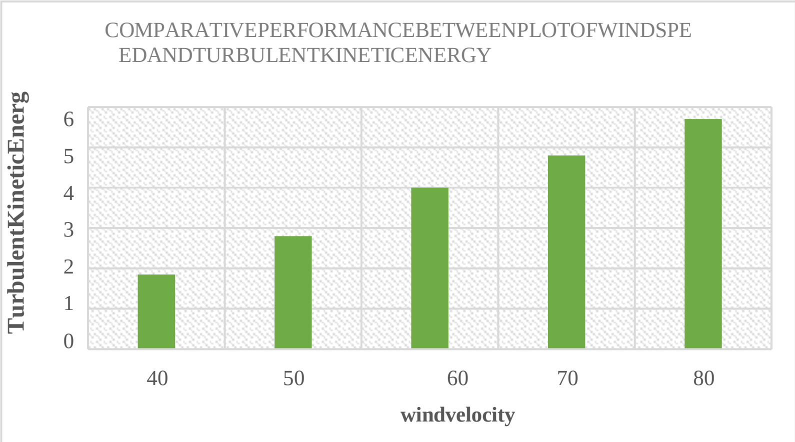

Figure33: ComparativeperformancebetweenPlotofWindspeedandturbulentkineticenergy.

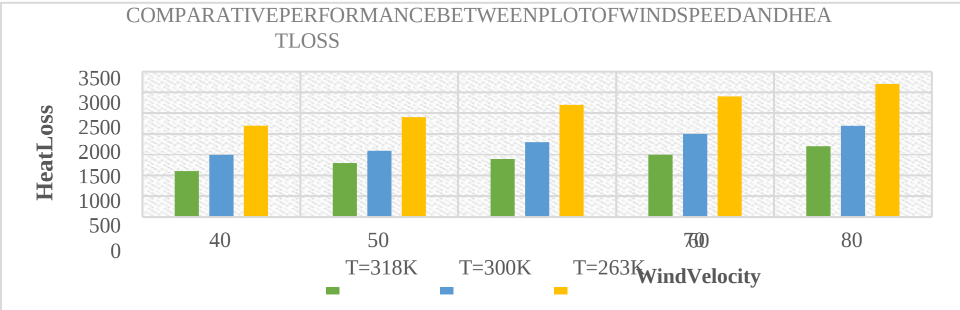

Figure 34: Comparative performance between Plot of Wind speed and Heat Loss.

VI CONCLUSIONS

The effect of surrounding environment temperature and wind velocity on heat transfer through the extended surfaces is analyzed with computational package. It is observed that the surface heat transfer coefficient increases as the wind velocity increases. Reduction in surrounding environmental temperature also results in excessive convective heat loss. The turbulence of wind flow increases with higher wind speed, which results in higher convection. The turbulent kinetic energy is also calculated. The empirical relation is derived between heat transfer coefficient and wind velocity: $\alpha = 6.9 \times U^{0.597}$. The temperature pattern and velocity pattern around the engine geometry can be visualized with CFD tool. The temperature contour can reveal the major areas where the thermal load is higher. The velocity cont our can help the designer to change fin design appropriately to increase the turbulence of flow.At a lower speed the air flow separated on the fin surface at the leeward side and the temperature on the fin surface increased there. To increase the cylinder cooling, the cylinder should have a greater number of fins. However, the cylinder cooling decreases with an increased number of fins if the fin spacing is too narrow. Because the air does not flow well between the fins, over lapping of thermal boundary layers occurs on the upper and lower fin surfaces. This results in less heat transfer coefficient. As fin spacing is increasing the heat transfer coefficient is also increasing. This reaches a limiting value. Further increase in fin spacing has less effect. The velocity cont our over the fins has revealed that, as the fin spacing is increasing there is better air distribution over the fin surface. As Fin Width increases, the heat transfer coefficient decreases up to a limiting value of 11 mm. After that, further increase in fin width has less effect. At less fin width, the average surface velocity of air is higher. The air is better distributed over the fin surface, this gives higher heat transfer coefficient. Further increment in fin width shows reduction in heat transfer coefficient because of restriction of flow over the entire fin surface.

References

1. *Raman Kumar, VijaykantPandey, "CFD Analysis of an IC Engine with and without groove" International Journal for Research in Applied Science & Engineering Tecnology, Volume 10 November 2022.*
2. *SujanShrestha*, Nitish Kumar Yadav, SumanBikram Bam. "Considering heat transfer through fins in IC motors" Researchgate.publication, August 2019*
3. *NamanSahu*, Vishal Gupta, Pradeep Kr. Kurmi "Thermal Analysis of Engine Blades Using FEM: A Review" International Journal of Technical Innovation in Engineering and Modern Sciences (IJTIMES) Volume 4, Issue 8, August-2018*

4. *Charan, Srivastav, Bharadwaj, "Thermal analysis on rectangular plate fin with holes using Ansys", International Journal of Creative Research Thoughts, 2018.*
5. *K.Rama Chandra Manohar, Yakkala.M.KRaghunadh, Somanath.B, Santosh, B. Koteswararao "Optimization of engine fins for different heat transfer and thermal conductivity" IOP Conf. Arr.: Materials Science and Engineering (2018)*
6. *KiranBeldar, AvinashPatil, "Design and Analysis of a Longitudinally Balanced Chamber with Rectangular Indentation", International Journal of Scientific Development and Research (IJSDR), 2017.*
7. *Rajesh, Rahamathullah, MalleswaraRao, "Planning and rationalization of motor housing blades with different geometries and materials in hot inspection", International Journal of Nuclear Engineering and Management, 2017.*
8. *Mayankjain, MahendraSankhala, KanhaiyaPatidar, "Blade Heat Transfer Inspection and Rationalization Dee Shape Change", International Journal of Mechanical and Manufacturing Engineering, Volume 5, Number 7, July 2017.*
9. *Komita, Reddy, "Ceramic Analysis of Chamber Disturbances Using Different Material Balances Using ANSYS," ICAAMM, Elsevier, 2016.*
10. *S. Ravikumar, Chandra, Harish, "Exploratory and Transient Thermal Analysis of Heat Sink Fins for CPU Processors for Better Performance", Materials Science and Engineering, 197, 2017.*
11. *Sandeep Kumar, NitinDubey, "Thermal Investigation and Analysis of Heat Dissipation Rate of Single Cylinder SI Engine", IJEDR, Volume 5, Number 2, 2017.*
12. *Mogaji, Owoseni, "Numerical Analysis of Radiation Effects on Heat Flow Through Rectangular Profile Heat Flow", American Journal of Engineering Research (AJER), Vol-6, No-10, pp-36-46, 2017.*
13. *Arefin, "Thermal analysis of modified pin heatsink for natural convection", 5th International Conference on Information Science, Electronics and Vision, 2015.*
14. *Balendra Singh, Satish Singh, "Research Paper on Heat Transfer in Notched Fin and UN Notched Fin", International Journal of Research in Applied Science and Engineering Technology (IJRASET), Volume 4, Issue 9, September 2016.*
15. *Kongre, Barde, "Review Paper on Thermal Analysis and Heat Transfer of Single Cylinder S. I. Motor Fins", International Journal of Engineering Research and Technology (IJERT), Volume 4, Number 30, 2016.*
16. *Natrayan, Selvaraj, Alagirisamy, "Worm Analysis of Engine Fins of Various Shapes" International Journal of Innovative Research in Science, Engineering and Technology, Volume 5, Number 5, May 2016.*
17. *Masao Yoshida, Soichi Ishihara, Yoshio Murakami, Kohei Nakashima and Masago Yamamoto"Air-CoolingEffectsofFinsonaMotorcycleEngine",JSME,2006,Vol.49,No.3, PP869-875.*
18. *K.Sathishkumar,K.Vignesh,N.Ugesh,P.B.SanjeevaprasathandS.Balamurugan"ComputationalAnalysisofHeatTransferthroughFinswithDifferentTypesofNotches",International Journal of Advanced Engineering Research and Science, 2017, Vol.4, Issue.2, PP175-183.*
19. *MohsinA.AliandS.MKherde"designmodificationandanalysisoftwo-wheelercoolingfins-a review", International Journal of Advances in Engineering & Technology, 2014, Vol. 7, Issue.3, PP998-1002.*
20. *S Chandra sekhar, P Satish Reddy and Ch.ChandraRao "Structural and Thermal Simulationof Fins Of An Air Cooled Engine Cylinder Under Varying Speed Conditions", InternationalJournalof Science Engineering and Advance Technology, 2014, Vol. 2, Issue 9, PP411-414.*
21. *G. Angel and N. SambasivaRao "Design and Analysis of Two-Wheeler Cooling Fins",InternationalJournalofEngineeringScienceandComputing,2016,Vol.6,Issue1,PP2373-2374.*
22. *GokulKarthik A and Dinesh R "Design Optimization and Heat Transfer Analysis ofEngineCylinderFinsbyVaryingMaterials",IJSRSET,2017, Vol.3,Issue3,PP55-59.*
23. *P.Harish, B.Ramakrishna Reddy, G.S.Md.WaseemAkram , K.MD. Hanief and KumairNaik"OptimizationofEngineCylinderFinsofVaryingGeometryandMaterial",InternationalJournalof Innovative Research inAdvanced Engineering, 2014, Vol.1Issue 2, PP67-73.*
24. *Prof. Arvind S. Sorathiya, Hiren P. Hirpara and Prof. Dr. P.P. Rathod "An Effect of DifferentParameters of Fins on Heat Transfer of IC Engine- Review Study", IOSR Journal of MechanicalandCivil Engineering, 2014, Vol. 11, Issue 3, PP63-71.*
25. *M.Rajesh, S.Rahamathullah and Dr. G Naga MalleswaraRao "Design And Optimization OfEngineCylinderFinsByVaryingGeometryAndMaterialWithThermalAnalysis",InternationalJournal of Core Engineering & Management, 2017, Vol. 4, Issue 1, PP424-432.*