IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526

International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering

Impact Factor 7.047 ⋇ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10466

Investigation Of Different Fins Across Engine By Using Experimental & CFD Simulation

Chopade Chetan Sunil[1], Chikane Vijay Balkrushna[2], Kokate Vivek Kisan[3]Chaudhari Nakul Satish[4], Prof. S.D.Wavale[5]

Student, Dept. of Mechanical Engineering, JSPMs ICOR Wagholi, Pune, Maharashtra, India

Assistant Professor, Dept. of Mechanical Engineering, JSPMs ICOR Wagholi, Pune, Maharashtra, India

Abstract: Nowadays, it is common knowledge in the car industry that in order to increase vehicle economy, heat dispersion throughout the engine must be naturally cooled. The heat transfer properties of this fin of various shapes will be researched using CFD simulations and fabrication of the best design to validate the results. Different fins of circular, triangular, and parabolic form are explored in this study to achieve the best heat dissipation shape for an engine. Fins are primarily used to improve the rate of heat transfer to the surrounding environment by boosting convection. Conduction, convection, and radiation all contribute to the quantity of heat transmission. The heat transfer rate rises as the temperature differential between the objects grows. Our project's major goal is to improve heat transfer rates using several types of fin (circular, triangular, and parabolic) arrangement geometry. CATIA software is used to create the 3D model. The results and conclusions will be drawn when the experimental testing is completed.

I. INTRODUCTION

Heat transfer is a subject of widespread interest to the student of engineering curriculum, practicing engineers & technicians engaged in the design, construction, testing and operation of the many diverse forms of heat exchange equipments required in our scientific and industrial technology. Electrical engineers apply their knowledge of heat transfer for the design of cooling systems for motors, generators & transformers. Chemical engineers are concerned with the evaporation, condensation, heating & cooling of fluids. An understanding of the laws of the heat transfer flow is important to Civil engineers in the construction of dams, structures and to the architect, in the design of buildings. The Mechanical engineer deals with problems of heat transfer, in the field of internal combustion engines, steam generation, refrigeration and heating & ventilation.

To estimate the cost, the feasibility and size of the equipment necessary to transfer a specified amount of heat in a given time, a detailed heat transfer analysis must be made. The dimensions of boilers, heaters, refrigerators and heat exchangers depend not only on the amount of heat to be transmitted but rather on the rate at which heat is to be transferred under given condition. The successful operation of equipment components such as turbine blades and walls of combustion chambers of gas turbine depends on the possibility of cooling certain metal parts by removing heat continuously at a rapid rate from the surface. These varied examples shows that in almost every branch of engineering, heat transfer problems are encountered, which cannot be solved by thermodynamic reasoning alone but required an analysis based on science of heat transfer

In Engine when fuel is singed heat is created. Extra heat is additionally produced by contact between the moving parts. Only approximately 30% of the vitality discharged is changed over into valuable work. The staying (70%) must be expelled from the engine to keep the parts from softening. For this reason, Engine have cooling instrument in engine to expel this heat from the engine. Some substantial vehicles utilize water-cooling framework and practically every one of the bikes utilizes Air cooled engine, since Air-cooled engine are just alternative because of certain favorable circumstances like lighter weight and lesser space prerequisite. The heat created during burning in IC engine ought to be kept up at more significant level to build warm productivity, yet to forestall the warm harm some heat should expel from the engine. In air cooled engine, expanded surfaces called balances are given at the outskirts of engine chamber to build heat move rate. That is the reason the examination of balance is critical to expand the heat move rate. Computational Fluid Dynamic (CFD) investigation has demonstrated upgrades in balance productivity by changing fin geometry, S fin pitch, number of balances and balance material. The issue of this part is it commitment of heat move coefficient (h), Temperature (T), Velocity, Fin Geometry (thickness, hole), material and surface harshness. Heat move rate that contribute while running the engine relies on the speed of vehicle, fin geometry and the encompassing temperature. On the off chance that the cooling rate diminishes, it brings about overheating prompting seizure of the engine. Simultaneously, an

DOI: 10.17148/IJIREEICE.2022.10466

expansion in cooling rate influences the turning over of the engine. In investigation of heat move, a fin is a surface that it is reaches out from an article to build the pace of heat move to or from the earth by expanding convection. The measure of conduction, convection or radiation of an article decides the measure of heat move. Expanding the temperature contrast between the item and the earth, expanding the convection heat move coefficient, or expanding the surface territory of the article builds the heat move. Some of the time it isn't efficient or it isn't practical to change the initial two choices. Adding a fin to an item, in any case, builds the surface region and can here and there be a practical answer for heat move issues.

are manifold: Besides cost related aspects one of the main advantages is the unmatched flexibility regarding integration in an existing or changing environment. Vehicles are the central elements of an AGVS as they perform the actual transportation tasks. The vehicles have to be designed individually according to the specific conditions of the environment they are used in [1]. This concerns load handling equipment, the navigation system, the drive configuration and other aspects.

II. LITERATURE SURVEY

1. Computational Analysis of Heat Transfer through Fins with Different Types of Notches, K. Sathishkumar, K. Vignesh, N. Ugesh, P. B. Sanjeevaprasath, S.

In this paper it represents the Engine as one of the important components in an automobile which is subjected to high temperature and thermal stresses. In order to cool the engine, the fins are another component which are used to dissipate the heat from the Engine. Fins are generally used to increase the heat transfer rate from the system to the surroundings. By doing computational flow analysis on the engine cooling fins, it is helpful to know about the heat dissipation rate and the Principle implemented in this project is to increase the heat transfer rate, so in this analysis, the fins are modified by putting different types of notches and are of same material. The knowledge of efficiency and effectiveness of the fins are necessary for proper designing of fins. The main objective of paper is to perform analysis to determine the flow of heat at various notches available and the analysis is done by using ANSYS – CFD Fluent software. The fins with various configurations were modelled using CREO 2.0 and analyses are done by using CFD – Fluent in order to find out the heat transfer rate. It is clear that the results from software and theoretically says that the fins with rectangular notch have greater heat transfer rate compared to that of the fins without holes, fins with holes and V shaped fins. Since the heat dissipation rate is more in rectangular notch so we conclude that the rectangular notch fins are most efficiency and best heat transfer notch among all types of notch.

2. Analysis of Varying Geometry Structures of Fins using Radiators K. Chinnarasu1, M. Ranjithkumar, P. Lakshmanan, K. B. Hariharan, N. K. Vigneshwaran and S. Karan

In this literature it presents as radiators are heat exchangers used to transfer thermal energy from one medium to another medium. In the existing plain fins type radiator are commonly used, which are usually set up in a cross flow arrangement made up of aluminum and copper alloy. Powerful fan and water pump is accompanied in this to greatly improve heat dissipation rate. For higher cooling capacity of radiator, addition of fins is one of the approaches to increase the cooling rate of the radiator. This method follows the principle of increasing contact surface. Contact surface can also be increased by varying fin geometrical structure. In this project simple modification has been carried out in the existing fin geometry with a view to improve its heat dissipated rate. The varying fin structures are Box type, Sharp type, Round type. Sharp type radiator fins is fabricated to evaluate the effectiveness of the radiator. Also comparison of conventional coolant with SiC Nano fluid has been carried out by using Solid works and Ansys software. The investigation of the automotive radiator with Nano fluid and water as a coolant in dissimilar geometrical structure of fin is effectively carried. From simulation of the radiator fins with conventional coolant (water) as coolant it is found that heat drop from 375K to 362.1K
i.e. 3.47% for round type fin followed by box type fin from 375K to 364.01K i.e. 2.94%, sharp type fin from 375K to 365.51K i.e. 2.54% and normal type fin from 375K to 368.07K i.e. 1.85%. Round type fin show high temperature drops in the simulation. From simulation of the radiator fins with SiC Nanofluid as coolant it is found that heat drop from 375K to 349.44K
i.e. 6.82% for round type fin followed by box type fin from 375K to 35364K i.e. 5.7%, sharp type fin from 375K to 354.59K 5.44% and normal type fin from 375K to 360.67K i.e. 3.83%. Round type fin show high temperature drops in the simulation. From both the above simulation it is found out that temperature drop is high in radiator with large contact surface that is round type fin and further this heat dissipation can be increased by application of Nanofluid as coolant.

3. **Finite Element Thermo-Structural Methodology for Investigating Diesel Engine Pistons with Thermal Barrier Coating, Paolo Baldissera, Cristiana Delprete, Politecnico di Torino**

In this paper it presents an combustion engine applications, metallic materials have been widely employed due to their properties castability and machinability with accurate dimensional tolerances, good mechanical strength even at high

IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526

International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering

Impact Factor 7.047 ⁜ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10466

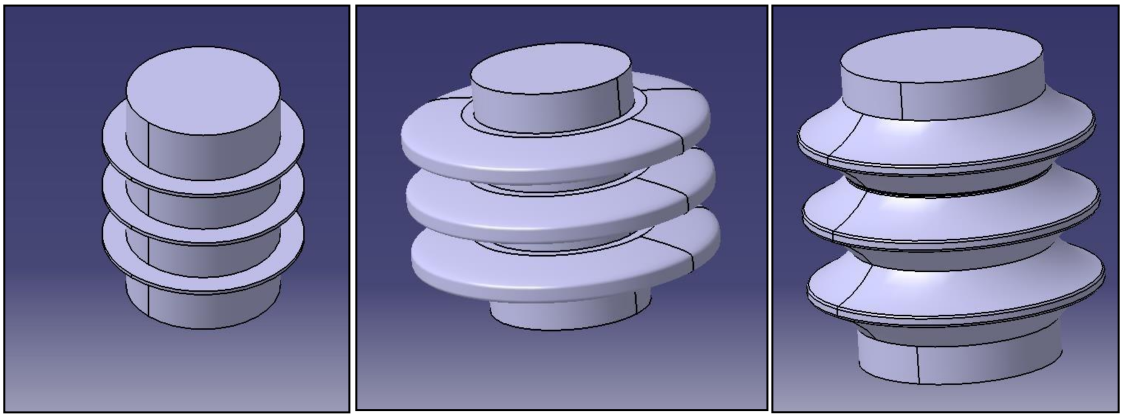
temperatures, wear resistance, and affordable price. However, the high thermal conductivity of metallic materials is responsible for consistent losses of thermal energy and has a strong influence on pollutant emission. A possible approach for reducing the thermal exchange requires the use of thermal barrier coating (TBC) made by materials with low thermal conductivity and good thermo-mechanical strength. In this work, the effects of a ceramic coating for thermal insulation of the piston crown of a car diesel engine are investigated through a numerical methodology based on finite element analysis. The study is developed by considering firstly a thermal analysis and then a thermo- structural analysis of the component. The loads acting on the piston are considered both separately and combined to achieve a better understanding of their mutual interaction and of the coating effect on the stress state. The thermal analysis pointed out a decrease of temperature up to 40°C in the upper part of the piston for the coated model. Despite the lower deformations induced by the reduced thermal load, the stiffening effect provided by the TBC results in higher peak stress. However, the lower temperature field inside the piston compensates by allowing higher yielding stresses for the component and reducing the impact on the safety factor. The thermal analysis has pointed out a decrease of temperature up to 40°C in the upper part of the piston for the coated model.

III. METHODOLOGY

- Step 1 :- Started the work of this project with literature survey. Gathered many research papers which are relevant to this topic. After going through these papers, we learnt about effect of fins with different shape arrangement in Engine.
- Step2 :- After that the selection of fins with different shape arrangement in Engine which are required for our project are decided.
- Step 3 :- After deciding the components, the 3D Model and drafting will be done with the help of CATIA software.
- Step 4 :- Computational Fluid Dynamics (CFD) simulations of muffler will be done with the help of ANSYS Fluent software.
- Step 5 :- The Experimental Testing will be carried out on optimized design from CFD simulation.
- Step 6 :- Comparative analysis between the experimental & CFD simulation.

IV. SYSTEM DESIGN

CAD:



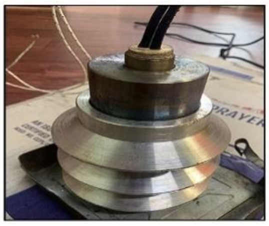
Computer-aided design (CAD) is the use of computer systems (or workstations) to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. The term CADD (for Computer Aided Design and Drafting) is also used.
Its use in designing electronic systems is known as electronic design automation (EDA). In mechanical design it is known as mechanical design automation (MDA) or computer- aided drafting (CAD), which includes the process of creating a technical drawing with the use of computer software.
CAD software for mechanical design uses either vector-based graphics to depict the objects of traditional drafting, or may

IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526
International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering
Impact Factor 7.047 ⁜ Vol. 10, Issue 4, April 2022
DOI: 10.17148/IJIREEICE.2022.10466

also produce raster graphics showing the overall appearance of designed objects. However, it involves more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions.
CAD may be used to design curves and figures in two-dimensional (2D) space; or curves, surfaces, and solids in three-dimensional (3D) space.
CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding, and aerospace industries, industrial and architectural design, prosthetics, and many more. CAD is also widely used to produce computer animation for special effects in movies, advertising and technical manuals, often called DCC digital content creation. The modern ubiquity and power of computers means that even perfume bottles and shampoo dispensers are designed using techniques unheard of by engineers of the 1960s

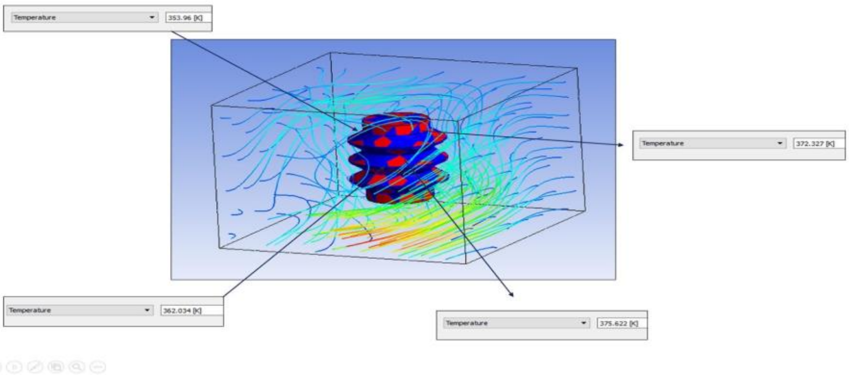
V. EXPERIMENTAL PROCEDURE

- Initially CFD simulation is performed to obtain optimum and most efficient shape for listed objectives from circular, triangular and parabolic shape fins around engine.
- It is observed from CFD simulation that parabolic fin has been most efficient design for present research and with greater heat dissipation effect along with standard design dimension material are final for manufacturing and testing purpose.
- Selection of standard bar with dimension of parabolic fin is machined under lathe machine to obtain final shape.
- Heater is inserted in inner surface of design to maintain constant specified temperature and blower is used to provide air effect in contact with fin.
- Due to heating of inner surface of model it starts to dissipate heat at faster rate as per parabolic fin design along with air blowing over the surface.
- Thermocouple are used to read temperature at fin surface and validation of result is performed with temperature obtained by CFD simulation probed at particular location

Impact Factor 7.047 ⋟ Vol. 10, Issue 4, April 2022
DOI: 10.17148/IJIREEICE.2022.10466

Fig. Experimental testing result



CONCLUSIONS

In present study CFD simulation of different fins are performed to obtain optimized model. Initially CFD simulation is performed to obtain optimum and most efficient shape for listed objectives form circular, triangular and parabolic shape fins around engine. It is observed from CFD simulation that parabolic fin has been most efficient design for present research and with greater heat dissipation effect along with standard design dimension material are final for manufacturing and testing purpose. It is observed from study that fins with parabolic shape have maximum heat flux and heat transfer coefficient compared to circular and triangular. In parabolic shape due to its shape, it has maximum heat transfer area across its surface. It is observed from CFD simulation result that temperature is around 89, 102, 80 and 90 degree Celsius. From experimental testing for similar boundary condition it is observed around 90, 97 and 102 degree Celsius. It is concluded that CFD and experimental result temperature distribution are similar in range and successfully validated. So, in further design research parabolic shape fin can be utilized for better heat dissipation and fast cooling rate shape design.

VI. FUTURE SCOPE

In the present work, we are altering the cross-sectional area of fins and accordingly we are noting temperature distribution. In future, we can study about different materials that affect the temperature distribution inside engine chamber. We can take case study of different material such as Copper, different Aluminum grades, composite material etc. and obtain its effect on temperature distribution of engine fins.

There is wide scope for future scholars to explore this area of research. Currently we have dealt with static analysis on fins across engine. This work can be further extended to make dynamic analysis on actual prototype testing model such as vibration analysis, thermal analysis, harmonic analysis etc.

In future, we can manufacture prototype model and implement it in two wheeled vehicles, four-wheeler engine fins and electrical vehicle system too.

DOI: 10.17148/IJIREEICE.2022.10466

For future work, we can perform case study of fins on reciprocating compressors, condensers on refrigeration system, electric transformers and motors, other electric and electronic components etc.

VII. REFRENCES

1. Computational Analysis of Heat Transfer through Fins with Different Types of Notches, K. Sathishkumar, K. Vignesh, N. Ugesh, P. B. Sanjeevaprasath, S. Balamurugan, International Journal of Advanced Engineering Research and Science (IJAERS) [Vol-4, Issue-2, Feb- 2017], ISSN: 2349-6495(P) | 2456-1908(O)
2. Finite Element Thermo-Structural Methodology for Investigating Diesel Engine Pistons with Thermal Barrier Coating, Paolo Baldissera, Cristiana Delprete, Politecnico di Torino, Italy, SAE International by SAE MOBILUS Open Platform, Thursday, January 10, 2019
3. J.-J. Shu, I. Pop, "Thermal interaction between free convection and forced convection along a vertical conducting wall, "International Journal of Heat and Mass Transfer
4. Wei Du, Lei Luo, Songtao Wang, Xinghong Zhang 'Effect of the dimple location and rotating number on the heat transfer and flow structure in a pin finned channel', "International Journal of Heat and Mass Transfer 127 (2018) 111–129"Published on 13 August 2018
5. Heat Transfer Analysis by CFD Simulation for Different shapes of Fins, Mohsin A. Ali and Prof. (Dr.) S.M Kherde, International Journal for Research in Applied Science & Engineering Technology (IJRASET), Volume 3 Issue II, February 2015 ISSN: 2321-9653
6. Design Modification and Analysis of Two Wheeler Engine Cooling Fin, Rashin Nath.KK, Jayee K Varghese, International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE) Vol 2, Issue 7, July 2017, ISSN 2456-1290
7. Analysis of I.C. Engine Fins for Effective Cooling Performance, Akash M Vyas Ruchir Parikh, IJSTE - International Journal of Science Technology & Engineering | Volume 5 | Issue 1 | July 2018 ISSN 2349-784