To Investigate Optimum Design of Fins Model for Better Heat Transfer by using ANSYS Software

Ambrish Pandey[1], Vikash Dwivedi[2]

[1]*M.Tech (Mechanical Engineering) scholar, B.N. College of Engineering & Technology Lucknow, U.P. (India)*
[2]*Assistant Professor, B.N. College of Engineering & Technology Lucknow, U.P. (India)*

Abstract - *Basically, the running of vehicles depends on the performance of engine.Currently, many industries facing problem of overheating in machine components due to heat generation within them. The manufacture made the appliances with compact design and low cost. But the heat needs to be transfer at higher rate to maintain the temperature of the machine components so that the component temperature remains within working temperature range. To overcome the problem of overheating, especially in thermal systems, fins are usually provided. Fins can be analyzed in design phase only using Computational Fluid Dynamics as tool and assuming uniform heat transfer coefficient model on its surface. Thermal inspection is a piece of material science which questions the properties of materials which undergo thermal inspection and which are subject to temperature change in addition. Heat transfer through structures such como internal ignition engines, moulding blocks and other applications is studied by thermal analysis in conduction and convection modes during heat exchange.*

The thermal structures must be designed and evaluated to make, disperse the reasonable proportion of undesirable heat with the required interest. The effective working of thermal hardware's based upon different elements, significantly cooling or heating of its specific parts. The issue emerges when the heat exchanged by these fins are insufficiently adequate to cool the heat producing devices and harm the parts of the devices. The fundamental point of the task is to advance the thermal properties of fins by various designs of geometry and enhance heat exchange rate

Key Words:* *Fins, Finite elemet method, Ansys, Modeling of fins, Thermal analysis of fins.

1. INTRODUCTION

The main effort of the study is to increase the heat transfer rate of fin which could be achieved by modifying certain parameters and geometry of the same. Fins are normally investigated by expecting uniform heat exchange coefficient design on its surface. In any case, optimization by different researchers uncovered that it isn't steady, however shifts along the fin length. It is basically a direct result of non- uniform obstruction experienced by the fluid flow in the between fin area. Expanding the heat exchange region heat dissipation rate enhances, yet the expansion of resistance to fluid flow causing a decrease in heat exchange. With a specific end goal to the dissipation of heat of high heat flux densities, the required heat sink should regularly be higher than devices. Subsequently, the heat sink execution is decreased.

The inner fin resistance might be reduced by including the notches or by adding the perforation to the fins. Adding a cross-fin in the center expands the heat transfer territory, however it forms the stagnant layer of hot air at the fin base. The fluid flow movement at the underside of the fin exhibit can be enhanced by adding holes to the fins.

1.1 Extended Surface (Fins)

A fin is a surface that increases from one component to increase the amount of heat transfer from or to the surrounding convection. Increasing the difference in temperature between the object and the atmosphere, increasing the convective heat transfer coefficient or increasing the surface of the component increases the thermal discharge. Sometimes the budget is not sufficient or the first alternatives can not be alternated. However, adding a fin to the item will increase the area of the surface and increase the heat transfer rate. In engineering programs, exceptional shape and length fins are used to improve the heat transfer rate including

- Rectangular fin
- Triangular fin
- Trapezium fins
- Circular segmental fins

Fins are utilized in a substantial number of engineering applications just to expand the heat exchange rate from surfaces. Typically, the fin comprises a blend of metal that has a high thermal conductivity. The fins are contacts in the open to a streaming liquid, which cools or warms the body, with the high thermal conductivity that permits to expanding heat exchange being led from the mass of the body through the blade. The shape and outlines of cooling balances is utilized in various circumstances.

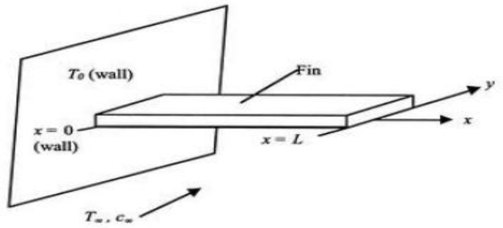

Fig-1.1: diagram of heat transfer fin

Heat transfer explains the thermal energy trading through heat dissipation between physical systems that rely on the weight and the temperature. Conduction or dispersion, convection and radiation are the basic methods of heat exchange. expect the territory an at models of various fins appeared in Figure 1.2 where warm is being exchanged from the surface at a settled temperature Ts to the encompassing liquid at a temperature $T\infty$ with a thermal exchange coefficient h. The heat exchange rate might be expanded by expanding the convection coefficient h, diminishing the liquid temperature $T\infty$, or adding materials to the territory A

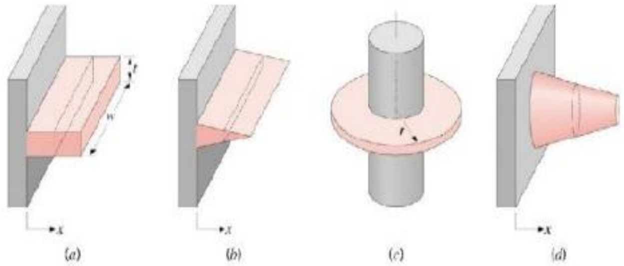

Fig-1.2: Use of extended surface or fin to enhance heat transfer.

1.1. Applications of Heat Sinks (Fins)

- Economizers for steam power plant
- Electrical transformers and motors
- Convectors for hot water and steam heating system
- Air-conditioned cylinders of aircraft engines, I.C. engines and air compressors
- Electronic equipment's

2. PREVIOUS WORK

Various researches carried out in past decade show that heat transfer through fins depend on number of fins, fin pitch, fin design, wind velocity, material and climate conditions. Different literature surveys suggest that variations in the cross-section influence how thermal transfer across extended surfaces and the thermal transfer coefficient.

Charan et. al. (2018) has analyzed expanded surfaces that are often used in a broad range of engineering applications to improve convection heat transfer. It has been discussed that the conception of introducing perforations on the lateral surface of fin is to enhance heat transfer rate effectively.

Sangaj et. al. (2018) Experimental research to determine the thermal distribution of various materials and geometries within the pin fine, the state heat transfer analysis was carried out with the software ANSYS for the testing and validation of outputs using a finite element. The main objective of the work is to optimize thermal properties through various geometries, materials and fine thicknesses. **Beldar et. al. (2017)**CFD software was used to perform steady state thermal analysis. Analysis of air flow, pressure drop analysis. The heat input varies by 10%, 20% and 30% by 25 Watts, 45 Watt and 65 Watts. The heat is different. In the sector, a fin array not being compensated, even though the heat transfer region decreases. After an entry on the middle part of the small, natural air flow decreases, the air velocity increases over the channel, the air pressure rises across the channel, the air temperature in the cylindrical heat sink increases. Using experimental and CFD tests it found that the optimum height transition relative to the other end range is 20 percent of the rectangeable notch scale of the longitudinal fine.

Rajesh et. al. (2017) The thermal properties of the fine and the thickness of cylinder fines have been analysed by varying geometry, material (Cu and Al alloy 6082). By variations in the circular geometry and also by varying fine thickness for both geometries, the Fins models are created. Pro / Engineer &UniGraphics are the 3D modeling software used. Cylinder fins are analyzed thermally to determine the temperature variation distribution over time. The analyzes are done with ANSYS. With thermal analysis on the fins of the engine cylinders, the heat dissipation inside the cylinders is beneficial. The theory used in this paper is the use of the invisible working liquid to maximize heat dissipation by air.

Jain et. al. (2017) Heat dissipation of fins was analyzed by changing their geometry. Parametric fine models to forecast the transient thermal behaviour have been developed. There are different geometries including rectangular, circular, triangular and extensive fines after models are made. CREO Parametric 2.0 is the simulation framework used. ANSYS 14.5 is used to analyze this. Current material is usually Aluminum Alloy 204, the thermal conductivity of which is 110-150W / m-°C. Analyzes were carried out with aluminum alloy 6061, which is about 160-170W / m-oC heat conductivity higher.

Kummitha et. al. (2017) Research carried out to determine the best material that ensures the highest heat transfer rate during the combustion processes keep the motor in secure operation and with high strength, low weight. A passion pro bike cylinder block was considered and based on GAMBIT software as well as thermal analysis with ANSYS software.

Ravikumar et. al. (2017) Experimented with geometric variables to improve thermal performance and the design of the heat sink. This project used thermal testing to identify a cooling solution for a 5W CPU desktop. The design enabled the chassis to be cooled with heat sink attached to the CPU to properly cool the entire system. This study considered the configuration of cylinder pin circular fins and rectangular heat sinking fins with aluminum base plating and the regulation of processes for the CPU heat sink. The heat dissipation was further enhanced by an alternative model of thermal fins.

Sandeep Kumar et. al. (2017) The rate of heat transfer in the heating area IC engine was studied. For this transient thermal study, the actual design of the Bajaj discovers 125 CC single cylinders were carried out. Thermal transient analysis was conducted to optimize geometry parameters and boost heat transfer from the IC engine to ensure that the engine cylinder configuration was actual and proposed.

3. FINITE ELEMENT METHOD

Methods of finite components are a valuable tool for finding a numerical solution for various technical studies. The method is common enough for all materials at different borders and load conditions for any complex form or geometry.

The method came as a means of analyzing the strain in a complex airframe structure in the aerospace industries. It grew up and became the technology used in aircraft design for matrix evaluation. Both researchers and practitioners have received wide popularity in the technique. The beauty of finite element technology is that a body or a form could be separated into small finite-dimensional pieces known as finite factors. The particular structure is taken into account when assembled into a final set of joints known as nodes.

3.1. General Procedure of Finite Element Method

The general problems by the technique of the finite elements are explained step by step.

The step-by-step process can be defined as follows for static structural applications:

Step 1: - Design model explanation (domain). The primary step in the process of finite elements is to separate the result area structure into subdivisions or elements.

Step 2: **-** Selected the correct form of interpolation. Since the explanation of a difficult structure (field vary) cannot be accurately predicted under any specific load conditions, we assume that the unknown solution can produce an appropriate result within a component. The expected outcome must be straightforward and certain convergence requirements must be fulfilled.

Step 3: - The start of stiffness matrices and load vectors (feature matrices). With the supposed version of displacement, both equilibrium situations and the correct variation control should be applied to the stiffness material..

Step 4: **-** Assembly of element equations to achieve equilibrium.

The individual matrices of the elements of rigidity and loading vectors must be constructed correctly since the structure consists of a great number of finite elements as

$$[K]\varphi = P \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (1)$$

When [K] is denoted a mounted stiffness matrix, age is known as the knot-displacement vector and P is the full shape vector or nodal pressure [K]

Step 5: - System equation solution to detect displacement nodal values (subject variable). The standard balance equations must be changed in order to take account of the limiting conditions of the problem.

The vector ' т ' can be resolved very problem-free in linear problems. However, the response must be received in a series of ladder, each with the amendment of the [K] and β is the weight vector P, in the case of non-linear problems.

Step 6:-Detail strains and stresses are calculated. If requested, by utilizing the essential equations of stable or structural mechanics, the detail lines and stresses can be calculated. The phrases in brackets in the above steps apply the overall FEM step by step.

3.2. Procedure for ANSYS analysis

Static measurement of displacement, stresses, stresses and forces in structures or materials is used because hundreds do not result in significant inertia or damping. In response conditions, stable loading is assumed. Loading types that can be used for a static test include external forces and loads, regular domestic inertial forces, imposed (non-0) movement speed, temperatures (for thermal stress). A linear or non-linear static analysis may be available. We don't neglect linear static analysis in our current work.

These key steps are the protocol for static analysis:

- Construction the model.
- Achieving the solution.
- Evaluation the results.

A structural examination is carried out using the ANSYS Workbench V.14.0 to carry out the finite element examination of the Fins models during the engine temperature transfer to the air with the aid of fins. At this stage research by the Fins involves a continuous thermal analysis and minor modifications in order to obtain designs. The fins model was built and saved in*.Igs in this file in Solidwork 2016 and imported to the workbench in ANSYS.

3.3. Material properties

For static analysis, Young's modulus (EX) should be illustrated. We define mass residences inclusive of density (DENS) if we intend to apply inert loads (that involve gravity). Similarly, we outline the coefficient of thermal growth (ALPX) if we intend to use thermal centers (temperatures).

3.4. Find the solution

This step defines the type and alternatives of analysis, follows hundreds and causes the final element response. Three levels are required:

- Pre – processor step
- Solution step
- Post-processor step

4. DESIGN AND ANALYSIS OF FIN MODEL

The surface that extruded out from a base is known as a fin. Fins are used by extending convection rates to improve the rate of heat dissipation from or to the environment. An object's entire convection, conduction, or radiation selects the heat rate it transferred. It increments with the distinction of temperature between surroundings and the object, additionally expanding the convection coefficient of heat exchange, or expanding the surface region. But, increase of the area also causes increased resistance to the heat flow. Hence, coefficient of heat transfer is based on the total area (the base and fin surface area) which comes out to be less than that of the base.Experiments to discover changes in temperature inside fins in three types of geometrics were performed (plating fins, perforation of circulation and rectangular perforation) with the use of a finite-element ANSYS software to check and approve results. Temperature variations in various plate fin areas are evaluated by FEM and the results of plate fins in experimental Ansys are compared with. The principle used in this project is to expand the rate of heat discharge through the use of wind flow. The principal objective of the thesis is to boost thermal properties by mixing geometry, material and fine design.

4.1. Thermal Analysis of Fins Model in Ansys

Fins models designed with the material selection of aluminum alloy 1060 and three Fins model designed here i.e. plate Fins, Circular Perforation,pin fins with circular shape and draft pin fins. Fins model with plate fins and perforation in type of circular, pin fins with circular shape and draft pin fins for passing air through fins for maximum heat transfer. FEM analysis performed through Ansys. The practical use of finite element modeling is known as FEA which is best understood during the real problem solving. FEA has been broadly utilized by the automotive business. It is an extremely prominent instrument for configuration builds in the product development technique. It is imperative to comprehend the FEA basics and design technique, demonstrating systems, the inherent mistakes and their impacts on the nature of the outcomes to render FEA as an effective design tool. FEA is also used as a computational tool for carrying out engineering problem analyses.

Experimental Fins Models become analyzed through with ANSYS that is partner with engineering simulation commercially used software program package supplying a whole organization that extents the complete type of physics, offering proper to use to nearly several field of thermal engineering utilities that a design technique. The software package deal uses its gear to area a digital product through a product testing technique like checking out a Fins models below totally different loading cases before it will become an extensive element.

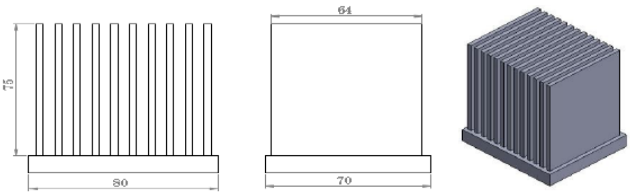

Fig- 4.1: Plate Fins Designed Model in Ansys

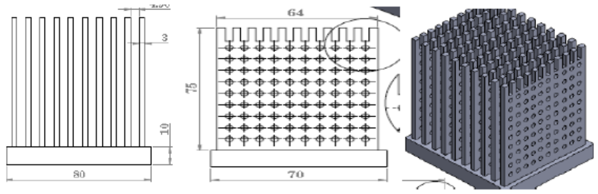

Fig- 4.2: CircularPerforation in plate fins designed Model in Ansys

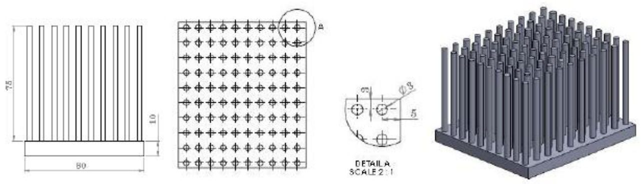

Fig- 4.3: Circular Pin Fins Designed Model in Ansys

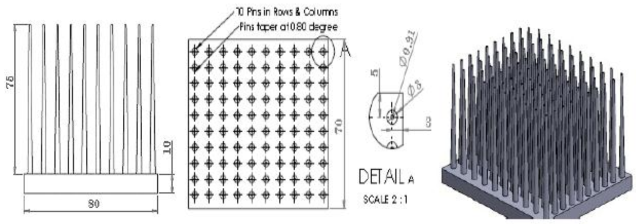

Fig- 4.4: Conical Draft pin finsdesigned Model in Ansys

4.2. Modeling of Fins

Modeling of the Fins done using Solid work has been explained in detail. The object of the investigation of finite elements is to recreate the mathematical behavior. The model includes all nodes, components, material properties, specific constants, limits and other characteristics used to describe the physical system. all models be generated then specific boundary conditions will be applied on the specific nodes then final analysis will be conducted.

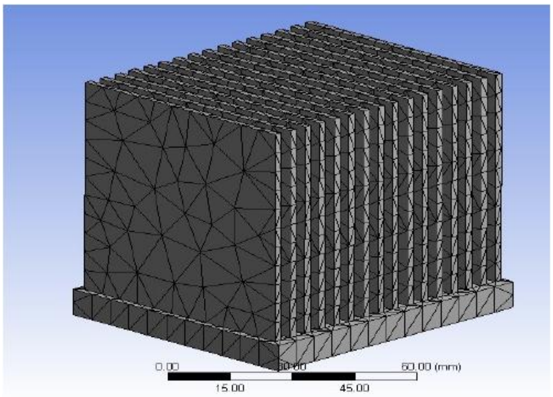

Fig- 4.5: Meshed Model of Fins without holes

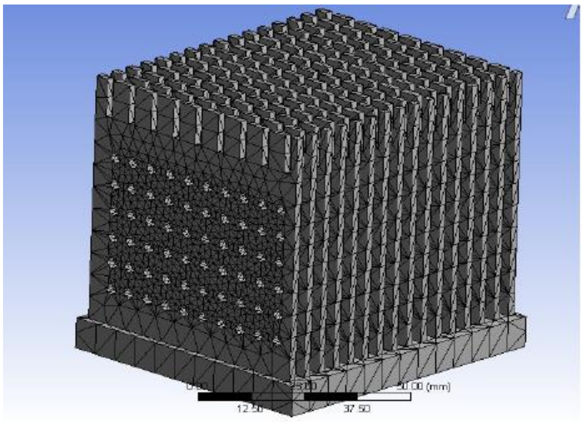

Fig-4.6: Meshed Model of Fins with Circular holes

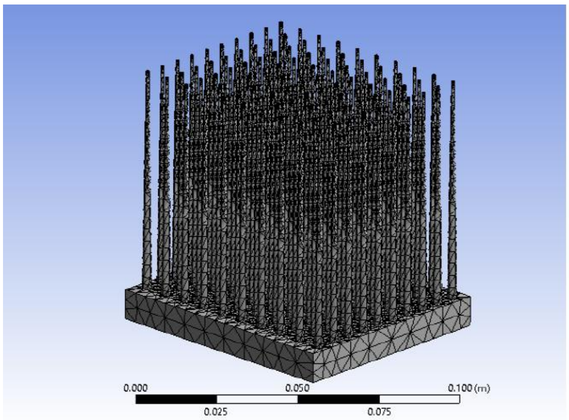

Fig- 4.7: Meshed Model of Draft Pin Fins

4.3. Material Properties

Thermal analysis of Fins performed by using Aluminum alloy of the Fins material, Composition of Aluminum alloy is shown in Table 4.2.

Table 4.2: Material properties of Fins Model

Parameters	Units	Aluminum alloy (1060)
Density	(Kg/m^3)	2700
Young's Modulus	(MPa)	69000
Coefficient of thermal expansion	(1/K)	2.3×10^{-5}
Poisson's Ratio	-	0.33
Elastic modulus	(GPa)	70
Ultimate Tensile Strength	(MPa)	310
Thermal conductivity	(W/m/^{0}C)	200

5. RESULT AND DISCUSSION

After designing the Fins model in Solid work, the. IGS FILE has been converted to IGES format. This configuration allows the design to be compatible in the ANSYS software. After importing the design in ANSYS, the process of analysis began. Mathematically, the Fins model to be investigates has been subdivided into a mesh of fine size elements. Analysis has been done for steady state condition and applied boundary conditions have been applied as shown in figures. After solver run, the Temperature and Total Heat Flux have been found out. These results have been obtained for three geometric variants i.e. fins without perforation, Fins with circular perforation and Fins with rectangular perforation.

5.1. Temperature distribution analysis of Fins Models

From thermal analysis of fins, maximum temperature observed at base is 47.534^{O}C which is for non-perforated fins and minimum temperature is observed for fins with square perforation which is 46.651^{O}C. Maximum Temperature drop is found in Circular perforated fins.

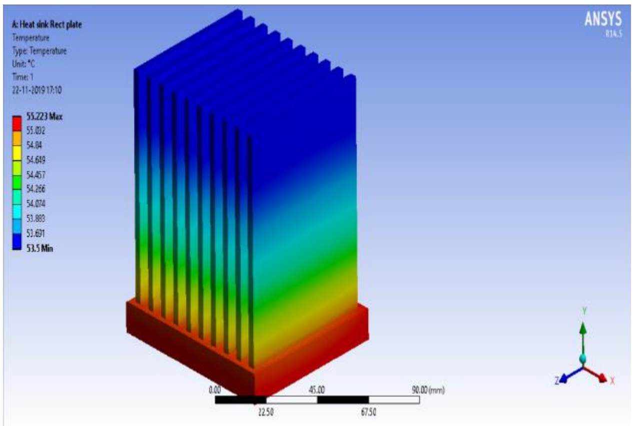

Fig- 5.1: Temperature Distribution of Plate Fins

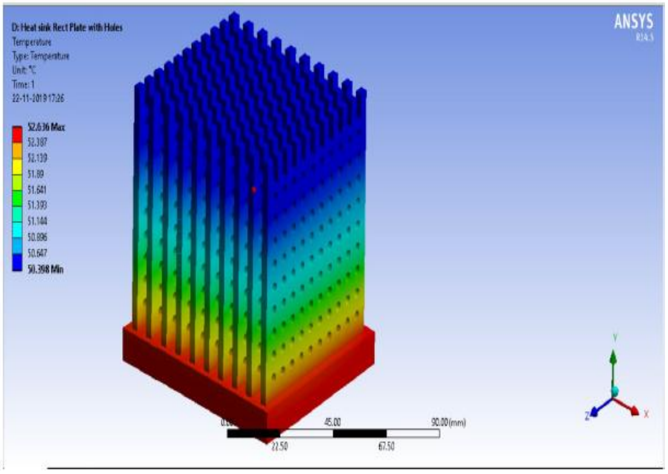

Figure 5.2: Temperature Distribution of Rectangular plate Fins with holes

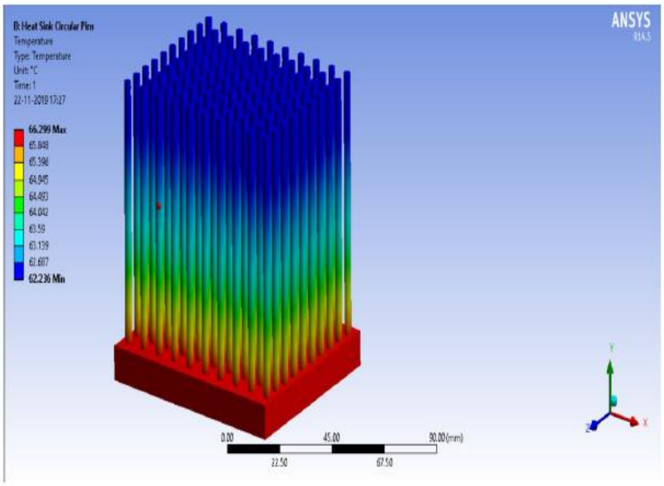

Figure 5.3: Temperature Distribution of Circular Pin Fins

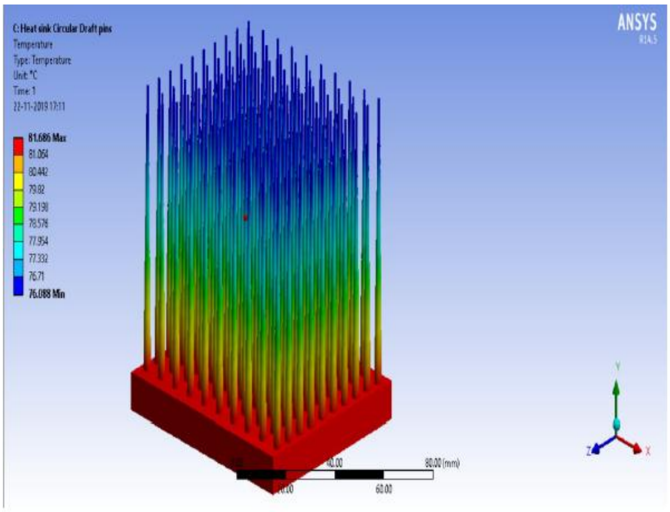

Figure 5.4: Temperature Distribution of draft conical Pin Fins

➢ **Heat Flux and thermal stress analysis of Fins Models**

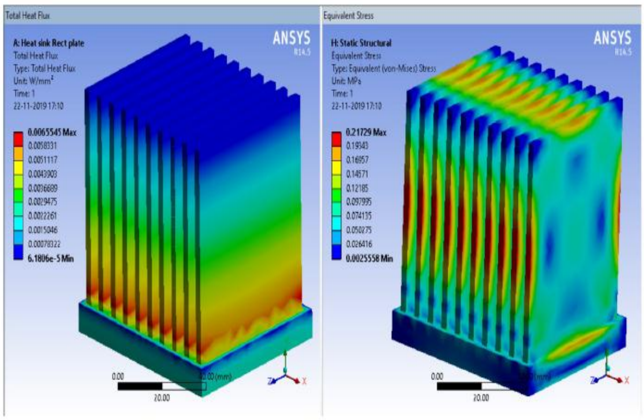

Figure 5.5: Heat Flux and Stress of Plate Fins

Table 5.1: Shows the Maximum and Minimum temperature variations

Geometry Condition	Heat Flow(watt)	Max Temperature (°C)	Min Temperature (°C)	Temperature drop(°C)
fins with rectangular plate	13	55.22	53.5	1.72
Circular pin fins	13	66.29	62.23	4.06
Conical draft pin fins	13	81.68	76.08	5.6
Rectangular plate fins with holes	13	52.63	50.39	2.24

Table 5.2: Heat Flux Found on All conditions of Fin Models

Geometry Condition	Heat Flux	Thermal Stress(MPa)	Weight(kg)
fins with rectangular plate	0.0065	0.217	0.554
Circular pin fins	0.016	0.073	0.301
Conical draft pin fins	0.015	0.074	0.223
Rectangular plate fins with holes	0.014	0.322	0.48

6. CONCLUSIONS

From the comparative analysis of plate Fins, Circular Pin fins, plate fins with holes, and draft Pin fins study conclusion is that the Total heat flux found maximum 0.016W/mm^{2} in Circular Pin fins. Maximum temperature found in Conical draft Pin Fins is 81.68^{0}C and Minimum Temperature found in Plate Fins is 55.32 ^{0}C. Maximum Temperature drop found on Conical draft pin fin 5.6 °C and minimum temperature drop found on plate fin is 1.72°C.

So, it is concluded that Pin Fins with Conical draft Pin Fins is shows better Heat transfer properties in this analysis. Thus, better heat transfer for fins or an experimental result shows that Conical draft Pin Fins having better than plate Fins. The overall analysis is performed on ANSYS FEM analysis Tool. Thus, further research can be carried with the advance materials and different designing, analysis tools. From the above study work the following conclusions are made:

- Thermal analysis for fins has been completed by adjusting other parameters, including geometry, plat fins, circular perforation in plate fins and pin fins.
- While looking at the results of the experiment, one can easily say that using Conical Pin Fins with aluminum alloy 1060 is much easier because of the dropping temperature and the heat transfer rate in Conical Pin Fins compared with Plate Pins

REFERENCES

[1] Charan, Srivastav, Bharadwaj, Thermal Analysis on Rectangular Plate Fin with Perforations Using Ansys, International Journal of Creative Research Thoughts, 2018.

[2] Sangaj, Shinde, Shanediwan, Potdar, Suryawanshi, 'Thermal and Parametric Analysis of Pin-Fin, International Research Journal of Engineering and Technology (IRJET), Volume 05, Issue 02, Feb-2018.

[3] Kiran Beldar, AvinashPatil Design and Analysis of cylinder having longitudinal fins with rectangular notches International Journal of Scientific Development and Research (IJSDR), 2017.

[4] Rajesh, Rahamathullah, Malleswara Rao, sign and optimization of engine cylinder fins by varying Geometry and material with thermal analysis International Journal of Core Engineering & Management, 2017.

[5] Mayank jain, MahendraSankhala, Kanhaiya Patida Heat transfer analysis and optimization of fins by Variation in geometry in the, International Journal of Mechanical and Production Engineering, Volume- 5, Issue-7, Jul.-2017.

[6] Kummitha, Reddy, 'Thermal Analysis of cylinder block with fins for different materials using ANSYS' ICAAMM, Elsevier, 2016.

[7] S.Ravikumar, Chandra, Harish, Experimental and Transient Thermal Analysis of Heat Sink Fin for CPU processor for better performance, Materials Science and Engineering, 197, 2017.

[8] Sandeep Kumar, Nitin Dubey, Investigation and Thermal Analysis of Heat Dissipation Rate of Single Cylinder SI Engine, IJEDR, Volume 5, Issue 2, 2017.

[9] Mogaji, Owoseni, Numerical Analysis of Radiation Effect on Heat Flow through Fin of Rectangular Profile, American Journal of Engineering Research (AJER), Volume-6, Issue-10, pp-36-46, 2017.

[10] Arefin, Thermal Analysis of Modified Pin Fin Heat Sink for Natural Convection in the 5th International Conference on Informatics, Electronics and Vision, 2016

[11] Balendra Singh, Satish Singh, "ortheA Research Paper on Heat Transfer in Notch Fin and UN Notch Fins, International Journal for Research in Applied Science & Engineering Technology (IJRASET), Volume 4 Issue IX, September 2016.