COMPARATIVE THERMAL ANALYSIS OF RECTANGULAR FINS USING ALUMINUM ALLOYS

A PROJECT REPORT SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENT FOR THE AWARD OF THE DEGREE OF BACHELOR OF TECHNOLOGY IN MECHANICAL ENGINEERING

SUBMITTED BY

ALLURI VISHNU	317126520002
DODDI GIRISH	317126520016
GORU KUSAL KUMAR	318126520L05
TEPPALA NAVEEN	317126520053
BORA SIVA VENKATA MRUNAL BABU	317126520008

UNDER THE ESTEEMED GUIDANCE OF

MRS BALA DIVYA POTNURU

ASSISTANT PROFESSOR

DEPARTMENT OF MECHANICAL ENGINEERING

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES
(AUTONOMOUS)

(Permanently Affiliated to Andhra University, Approved by AICTE, and Accredited by NBA & NAAC with ‘A’ grade) SANGIVALASA BHEEMUNIPATNAM (MANDAL) – 531162
VISAKHAPATNAM (DIST.), ANDHRA PRADESH, INDIA.

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (A)
(Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with A grade)
SANGIVALASA, VISAKHAPATNAM (District) – 531162

PRAGNANAM BRAHMA
ANITS

CERTIFICATE

This is to certify that the Project Report entitled **"COMPARATIVE THERMAL ANALYSIS OF RECTANGULAR FINS USING ALUMINUM ALLOYS"** being submitted by ALLURI VISHNU (317126520002), DODDI GIRISH (317126520016), GORU KUSAL KUMAR (318126520L05), TEPPALA NAVEEN (317126520053), BORA SIVA VENKATA MRUNAL BABU (317126520008) in partial fulfillments for the award of degree of **BACHELOR OF TECHNOLOGY** in **MECHANICAL ENGINEERING, ANITS**. It is the work of bona-fide, carried out under the guidance and supervision of **MRS.P BALA DIVYA**, Assistant Professor, Department Of Mechanical Engineering, ANITS during the academic year of 2017-2021.

PROJECT GUIDE

T. Bala Divya.

(MRS.P BALA DIVYA)
Assistant Professor
Mechanical Engineering Department
ANITS, Visakhapatnam.

Approved By
HEAD OF THE DEPARTMENT

(Dr. B. Naga Raju)
Head of the Department
Mechanical Engineering Department
ANITS, Visakhapatnam.

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
Sangivalasa-531 162 VISAKHAPATNAM Dist A.P

ACKNOWLEDGEMENT

We would like to express profound gratitude to our guide MRS BALA DIVYA POTNURU Assistant Professor, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences Sangivalasa, Bheemunipatnam Mandal, Visakhapatnam District for his inevitable support, encouragement, supervision and useful suggestions throughout this project, which enabled us to complete our work successfully.

We are grateful to Prof. T.V. HANUMANTHA RAO, Principal, Anil Neerukonda Institute of Technology & Sciences for his valuable suggestions.

We sincerely thank Dr. B. NAGA RAJU, HOD and Professor, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology and Sciences Sangivalasa, Bheemunipatnam Mandal, Visakhapatnam District for his valuable suggestions during the work.

We express our sincere thanks to the members of teaching and non-teaching staff of mechanical engineering for their kind co-operation and support to complete the project work.

Submitted by:

ALLURI VISHNU	**(317126520002)**
DODDI GIRISH	**(317126520016)**
GORU KUSAL KUMAR	**(318126520L05)**
TEPPALA NAVEEN	**(317126520053)**
MRUNAL BABU	**(317126520008)**

NOMENCLATURE

Q_i = Heat transfer, W/mm^2

K = Thermal conductivity, W/mK

dt/dx = Temperature gradient, K/m

m̧ = Mass flow rate, kg/s

C_p = Specific heat at constant pressure, J/kgK

P_r = Prandtl number

G_r = Grashof number

R_a = Rayleigh number

A_s = Surface area, m^2

T_s = Surface temperature, K

T_a = Ambient temperature, K

Latin symbols

ß = Coefficient of thermal expansion, 1/K

ƞ = Efficiency

ρ̇ = Density, kg/m^3

µ = Absolute viscosity, Ns/m^2

v = kinematic viscosity, m^2/s

LIST OF TABLES

LIST OF FIGURES

INDEX Page no

CHAPTER 1 INTRODUCTION

CHAPTER 2 LITERATURE REVIEW

CHAPTER 3 MODELLING OF FINS USING CATIA SOFTWARE

CHAPTER 4 ANALYSIS OF FINS USING ANSYS

CHAPTER 5 RESULTS AND DISCUSSIONS

CHAPTER 6 CONCLUSION AND FUTURE SCOPE

REFERENCES

ABSTRACT

In most of the engineering applications, heat dissipation is an important factor, which if not taken care of might result in high-thermal stresses and mechanical failures. So, the focus has been made mainly on heat dissipation and found out, the amount of heat dissipated depends upon the material and surface area which can be increased by the use of extended surfaces. And also the amount of heat it transfers increases by increasing the temperature difference between the fin configuration. The main aim of the present project is to analyze the performance of fin by varying material. Fin of rectangular configuration is designed with the help of CATIA V5R16 software. The analysis was carried out using ANSYS 15.0. The heat transfer performance of fin with same base, temperature of diffrent materials are compared. Comparative study has been done among the fin materials to find out the best material under steady state thermal conditions.

CHAPTER 1

INTRODUCTION

Fins are set on the surface of the cylinder to improve the quantity of heat exchange by convection. For example, When fuel is burned in an engine, heat is produced. Additional heat is also generated by friction between the moving parts. In air-cooled I.C engine, extended surfaces called fins are provided at the periphery of engine cylinder to increase heat transfer rate. That is why the analysis of fin is important to increase the heat transfer rate. The main of aim of this work is to study various researches done in past to improve heat transfer rate of cooling fins by changing cylinder fin geometry and material. In the present work, Experiments have been performed to discover the temperature variations inside the fins made in different geometries.

1.1 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 principle of conduction, convection, radiation of a fin configuration determines the amount of heat it transfers. Adding a fin configuration to the object, however, slightly increases the surface area, it can sometimes be economical solution to heat transfer problems. Circumferential fins around the cylinder, square and rectangular shape of a motor cycle engine and fins attached to condenser tubes of a refrigerator are a few familiar examples only occurs when there is a temperature difference, Flow faster when this difference is higher, always flows from high to low temperature, Is greater with greater surface area.

In I.C engines, out of the total energy produced inside the engine, approximately 30% of it is available at crankshaft in the form of propelling force & rest of the 70% of energy is exhausted in the atmosphere. The energy dissipation in the atmosphere is in the form of heat in the cooling water, exhaust gas, lubricating oil & through fins.

IC engine fins are the extended surfaces provided on the periphery of cylinder surface for heat dissipation. The heat transfer from engine to atmosphere through fin occurs by the mode of convection. To enhance the rate heat transfer by fin, the surface area must be increased. But due to space restriction it is not feasible.

So the aim of this project is to do few modifications in the fin geometry & shape in the fin design so as to achieve the optimum design of fin geometry which can help to improve heat dissipation.

1.2 Necessity of Cooling System

With the increase in heat dissipation from microelectronics devices and the reduction in overall form factors, thermal management becomes a more important element of electronic product design.

Heat sinks are devices that enhance heat dissipation from a hot surface, usually the case of a heat generating component, to a cooler ambient, usually air which is assumed to be the cooling fluid. In most situations, heat transfer across the interface between the solid surface and the coolant air is the least efficient within the system, and the solid-air interface represents the greatest barrier for heat dissipation. A heat sink lowers this barrier mainly by increasing the surface area that is in direct contact with the coolant. This allows more heat to be dissipated and/or lowers the device operating temperature. The primary purpose of a heat sink is to maintain the device temperature below the maximum allowable temperature specified by the device manufacturers.

Both the performance reliability and life expectancy of electronic equipment are inversely related to the component temperature of the equipment. The relationship between the reliability and the operating temperature of a typical silicon semi-conductor device shows that a reduction in the temperature corresponds to an exponential increase in the reliability and life expectancy of the device.

Reduction of temperature at the device will reduce stress in the component internal surface both the performance and reliability of electronic circuitry are strongly influenced by the temperature. Exposure to temperature beyond which the circuit is designed to withstand may result in failure of the circuit to perform to specification or in failure together. The maximum temperature to which the circuit will meet the electrical speciation with power allied, and the maximum storage temperature is defined as maximum temperature when the power is off, to high circuit may be exposed for a given period of time without the detrimental effects. The detrimental effects of excessive temperature may be divided into three categories as shown in the table 1.

Failure modes	Characteristics
Soft failures	Circuit continues to operate but does not meet specifications when the temperature is elevated beyond the maximum operating temperature. Circuit returns to normal operation when the temperature is lowered. Failure is due to change in component parameters with temperature.
Hard failures (Short Term)	Circuit does not operate. Circuit may or may not return to normal operation when the temperature is lowered. Failure is likely due to component but also may be due to change in component parameters with temperature.

Hard failures (Long Term)	Circuit does not operate at any temperature. Failures are irreversible. Failure may be caused by corrosion or inter metallic formation or similar phenomenon. Failures may also be caused by mechanical stress due to difference in temperature coefficient of expansion between a component and substrate.

Table 1: Characteristics of electronic cooling system

Soft failures happen as a result of the tendency of the parameters of both active and passive components to exhibit a degree of sensitivity to temperature.

As the temperature increases, the cumulative effects of component parameters drift may eventually cause the circuit output variables to deviate from the specification. Hard failures in the short term may occur as a result of component overload as a result of excessive heat or as a result of the breakdown of component attach or packaging materials.

Hard failures in the long term may occur for a variety of reasons such as corrosion, chemical reactions, and inter-metallic compound formation all of which are accelerated by elevated temperature. Hard failures may occur as a result of mechanical stress due to differences in the temperature coefficient of expansion between two materials joined together such as a component mounted on a circuit board.

In order to maximize circuit performance and reliability, it’s important to be able to predict the maximum operating and storage temperatures and to design and package electronic circuits, to minimize heat generated during operation and to minimize circuit temperature by employing adequate thermal management methods.

In selecting an appropriate heat sink that meets the required thermal criteria, one needs to examine various parameters that affect not only the heat sink performance itself, but also the overall performance of the system. The choice of a particular type of heat sink depends largely to the thermal budget allowed for the heat sink and external conditions surrounding the heat sink. It is to be emphasized that there can never be a single value of thermal resistance assigned to a given heat sink, since the thermal resistance varies with external cooling conditions.

When selecting a heat sink, it is necessary to classify the air flow as natural, low flow mixed, or high flow forced convection. Natural convection occurs when there is no externally induced flow and heat transfer release slowly on the free buoyant flow of air surrounding the heat sink. Forced convection occurs when the flow of air is induced by mechanical means, usually a fan or blower. There is no clear distinction on the flow velocity that separates the mixed and forced flow regimes. It is generally accepted in applications that the effect of buoyant force on the overall heat transfer diminishes to negligible level (under 5%) when the induced air flow velocity excess 1 to 2 m/s.

1.3 Determination of Volume of Heat Sink:

Table 2 shows approximate ranges of volumetric thermal resistance of a typical heat sink under different flow conditions.

Flow condition m/s	**Volumetric Resistance** cm^3 °C/W (in^3 °C/W)	
natural convection	500-800	(30-50)
1.0 (200)	150-250	(10-15)
2.5 (500)	80-150	(5-10)
5.0 (1000)	50-80	(3-5)

Table 2: Range of volumetric thermal resistance

The volume of a heat sink for a given low condition can be obtained by dividing the volumetric thermal resistance by the required thermal resistance. Table 2 is to be used only as a guide for estimation purposes in the beginning of the selection process. The actual resistance values may vary outside the above range depending on many additional parameters, such as actual dimensions of the heat sink, type of the heat sink, flow configuration, orientation, surface finish, altitude, etc. The smaller values shown above correspond to a heat sink volume of approximately 100 to 200 cm^3 (5 to 10 in^3) and the larger ones to roughly 1000 cm^3 (60 in^3).

Although there are many parameters to be considered in optimizing a heat sink, one of the most critical parameters is the fin density. In a planar fin heat sink, optimum fin spacing is strongly related to two parameters: flow velocity and fin length in the direction of the flow. The average performance of a typical heat sink is linearly proportional to the width of a heat sink inthe direction perpendicular to the flow and approximately proportional to the square root of the fin length in the direction parallel to the flow. For example, an increase in the width of a heat sink by a factor of two would increase the heat dissipation capability by a factor of two, whereas and increase the heat dissipation capability by a factor of 1.4. Therefore, if the choice is available, it is beneficial to increase the width of a heat sink rather than the length of the heat sink. Also, the effect of radiation heat transfer is very important in natural convection, as it can be responsible of up to 25% of the total heat dissipation. Unless the component is facing a hotter surface nearby, it is imperative to have the heat sink surfaces painted or anodized to enhance radiation.

If the cooler is insufficient and the temperature exceeds the maximum operating temperature, then this does not mean that the CPU is automatically damaged. With CPUs, we will usually encounter crashes if the CPU is overheated; but these go away as soon as the CPU is cooler again. In the long term, running the CPU at a temperature that is too high may reduce the CPU life, since an overheated CPU is more prone to electromigration - even if it runs stable. With P4 CPUs, the CPU will turn its speed down automatically when it overheats. No damage to the CPU is possible, but the system will get slower while it's hot (which, in some cases, users might not even notic

1.4 Heat sink

A heat sink is an environment or object that absorbs and dissipates heat from another object using thermal contact (either direct or radiant). Heat sinks are used in a wide range of applications wherever efficient heat dissipation is required; major examples include refrigeration, heat engines, cooling electronic devices and lasers.

A heat sink without a fan is called a passive heat sink; a heat sink with a fan is called an active heat sink. Heat sinks are generally made of an aluminum alloy and often have fins.

Heat is generated in many ways. Systems performing work give up their energy in the form of heat. Chemical reactions release potential energy in the form of heat. Electron flow through materials generates heat. Compression of gases generates heat. Radiation exciting matter creates heat. Heat is an energy associated with the random motion of atoms and molecules.

The amount of sensible heat present in a body manifests itself as the temperature of the body. As heat is added to a body, the temperature rises. More sensible heat per unit volume means a higher temperature.

The first law of thermodynamics states that the total heat input must equal the total heat output plus any heat energy stored or converted to other forms of energy in the system. The second law of thermodynamics further states that heat flows from a hotter to a colder body and never the other way around. These two laws are the foundation of heat transfer.

The three basic mechanisms of heat transfer are: Radiation, Conduction & Convection. In actual situations all three of these are taking place simultaneously.

It is apparent however that in each of the above process, heat is transformed from one area to another because of temperature difference. Heat always flows from hot to cooler area. The amount of heat flow is determined by the difference in temperature of a second area. This leads to the obvious conclusion that the low temperature area limits the cooling and the component temperatures must calculated relative to the ambient temperatures. It is also important to note that the temperature the amount of heat flow is proportional to the amount of exposed surface of the heated area as well as the heat conductivity of the transfer path. The fundamental heat transfer problem is electronics is the removal of internally developed heat by increasing heat conductivity between the heat sources and the ultimate sink. One of the most economical and satisfactory methods is the use of a metal heat sink, which by conduction, radiation and convection increases the dissipation of heat.

A heat sink is a structure that facilitates the drain of heat from a hotter body in contact with it. It is essential a thermal short circuit from the hot body to any readily available cooling fluid. The mechanical heat sink has this ability because it can transfer heat to the surroundings better than the hotter body alone can transfer the heat at the same temperature.

A good heat sink design tends to minimize temperature gradients in the heat sink and maximize surface area per unit volume consistent with the transported throughout the heat sink to the exposed surface.

Cooling technique employed heat generated at one point is readily transported throughout the heat sink to the exposed surface. The smaller the temperature drop is to the heat transfer surface, the more efficient the heat sink. The shape of the conduction path in the material affects the performance.

A large cross section aids heat flow while a thin cross section offers a great resistance to flow. The plane normal to the flow of heat ideally offers the greater cross-sectional area at points of heat input.

Heat sinks materials are selected for their ability to perform as extended surfaces. The material must be a good thermal conductor. Silver is the best metal conductor, but is much to expensive for most applications.

Copper is an excellent thermal conductor and find many applications where high conductivity is required. Copper has twice the conductivity of aluminum but has thrice the specific gravity.

Aluminum is another material widely used in heat transfer work. Its ease of fabrication and light weight make it excellent material for forming long individual fins.

1.5 Principle

Heat sinks function by efficiently transferring thermal energy ("heat") from an object at high temperature to a second object at a lower temperature with a much greater heat capacity. This rapid transfer of thermal energy quickly brings the first object into thermal equilibrium with the second, lowering the temperature of the first object, fulfilling the heat sink's role as a cooling device. Efficient function of a heat sink relies on rapid transfer of thermal energy from the first object to the heat sink, and the heat sink to the second object .

The most common design of a heat sink is a metal device with many fins. The high thermal conductivity of the metal combined with its large surface area result in the rapid transfer of thermal energy to the surrounding, cooler, air. This cools the heat sink and whatever it is in direct thermal contact with. Use of fluids (for example coolants in refrigeration) and thermal interface material (in cooling electronic devices) ensures good transfer of thermal energy to the heat sink. Similarly, a fan may improve the transfer of thermal energy from the heat sink to the air.

1.6 Characteristics

High heat sink surface It's at the surface of the heat sink where the thermal transfer takes place. Therefore, heat sinks should be designed to have a large surface; this goal can be reached by using many fine fins, or by increasing the size of the heat sink itself.

Good aerodynamics Heat sinks must be designed in a way that air can easily and quickly float through the cooler and reach all cooling fins. Especially heat sinks having a very large number of fine fins, with small distances between the fins may not allow good air flow. A compromise between high surface (many fins with small gaps between them) and good aerodynamics must be found. This also depends on the fan the heat sink is used with: A powerful fan can force air even through a heat sink with lots of fine fins with only small gaps for air flow - whereas on a passive heat sink, there should be fewer cooling fins with more space between them. Therefore, simply adding a fan to a large heat sink designed for fan less usage doesn't necessarily result in a good cooler.

Good thermal transfer within the heat sink Large cooling fins are pointless if the heat can't reach them, so the heat sink must be designed to allow good thermal transfer from the heat source to the fins. Thicker fins have better thermal conductivity; so again, a compromise between high surface (many thin fins) and good thermal transfer (thicker fins) must be found. Of course, the material used has a major influence on thermal transfer within the heat sink. Sometimes, heat pipes are used to lead the heat from the heat source to the parts of the fins that are further away from the heat source.

Perfect flatness of the contact area the part of the heat sink that is in contact with the heat source must be perfectly flat. A flat contact area allows you to use a thinner layer of thermal compound, which will reduce the thermal resistance between heat sink and heat source.

1.7 Performance

Heat sink performance (including free convection, forced convection, liquid cooled, and any combination thereof) is a function of material, geometry, and overall surface heat transfer coefficient. Generally, forced convection heat sink thermal performance is improved by increasing the thermal conductivity of the heat sink materials, increasing the surface area (usually by adding extended surfaces, such as fins or foam metal) and by increasing the overall area heat transfer coefficient (usually by increase fluid velocity, such as adding fans, pumps, etc.)

For more complex heat sink geometries, and/or heat sinks with multiple materials, and/or heat sinks with multiple fluids, computation fluid dynamics (CFD) analysis is recommended.

1.8 Components of Heat Sinks

- Base Plate
- Fins or Extendable surfaces

1.8.1 Base plate

Base plate is part of heat sink on which the component is mounted. The base plate should be a good conductor of heat. It can be made of material, which has good thermal conductivity.

The base plate should have minimum thickness so that riveting of screwing of the component on to the base plate is possible.

If fins are machined as integral part of the base plate from which they are extend, there is no contact resistance at the base. However, more commonly, fins are manufactured separately and are attached to the base plate by a metallurgical or adhesive joint.

Alternatively, the attachment may involve press fit, for which the fins are forced into slot machines on the base plate material. In such cases, there a thermal contact resistance, which may adhesively affect the overall performance.

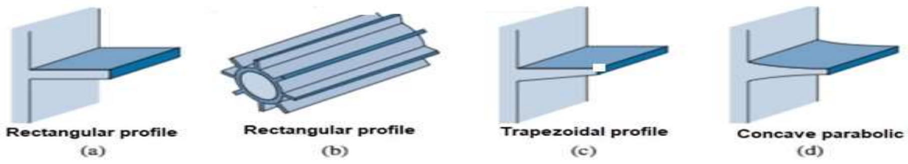
1.9 Types of fins

Fins can be broadly classified as:

1. Longitudinal fin
2. Radial fin
3. Pin fin

1)Longitudinal fin

Different types of Longitudinal fins are in Fig1.9.1.



Rectangular profile (a)

Rectangular profile (b)

Trapezoidal profile (c)

Concave parabolic (d)

Fig1.9.1 Different types of Longitudinal fins

(a) Longitudinal fin – Rectangular profile
(b) Longitudinal fin – Rectangular profile
(c) Longitudinal fin – Trapezoidal profile
(d) Longitudinal fin – Concave parabolic

2) Radial fin

e) Radial fin – Rectangular profile
(f) Radial fin – Triangular profile

Different types of Radial fins are shown in Fig1.9.2.

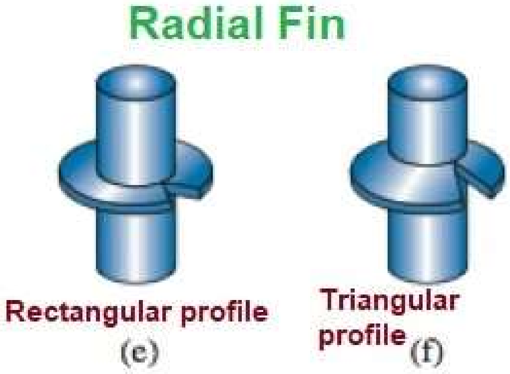


Fig1.9.2 Different types of Radial fins

3) Pin fin

(g) Pin fin – Cylindrical
(h) Pin fin – Tapered profile
(i) Pin fin – Concave parabolic

Different types of Pin fin shown in Fig1.9.3

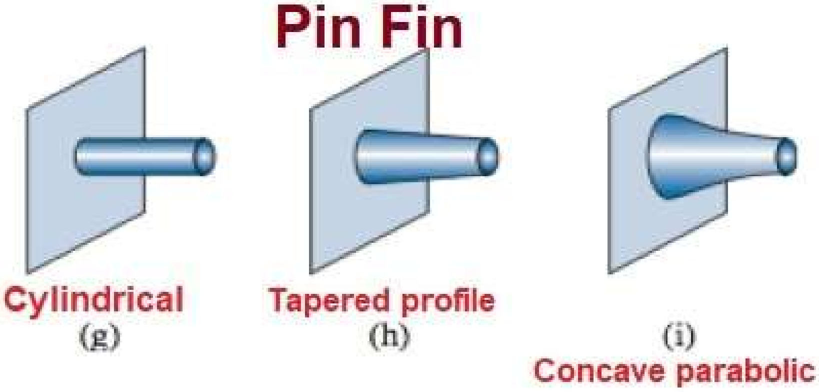
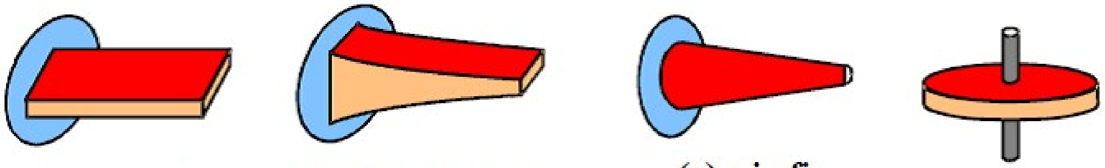


Fig1.9.3 Different types of Pin fins

Other types of Fins are:

- Constant area straight fin
- Variable area straight fin
- Pin Fin
- Annular Fin

Other types of fins are shown in Fig1.9.4.



(a) constant area straight fin (b) variable area straight fin (c) pin fin (d) annular fin

Fig1.9.4 Other types of fins

1.10 Materials of fins

- The most common heat sink materials are aluminum alloy has one of the higher thermal conductivity values at 229 W/mK but is mechanically soft.
- Copper has around twice the thermal conductivity of aluminum and faster, more efficient heat absorption. But it is more expensive than aluminum.

1.11 Heat Transfer From Extended Surface

The rate of heat transfer from a surface at a temperature Ts to the surrounding medium at T infinity is given by Newton's law of cooling as,

$$\dot{Q}_{\text{conv}} = hA_s(T_s - T_\infty)$$

Equation of fins

Where,
As is the heat transfer surface area and where.

h is the convection heat transfer coefficient

Temperatures Ts and Tinfinity are fixed by design considerations,

There are two ways to increase the rate of heat transfer:

To increase the convection heat transfer coefficient (h) – Increasing h may require the installation of a pump or fan, or replacing the existing one with a larger one, but this approach may or may not be practical. Besides, it may not be adequate.

To increase the surface area (As) – The alternative is to increase the surface area by attaching to the surface extended surfaces called fins made of highly conductive materials such as aluminum. Finned surfaces are manufactured by extruding, welding, or wrapping a thin metal sheet on a surface. Fins

enhance heat transfer from a surface by exposing a larger surface area to convection and radiation.

Finned surfaces are commonly used in practice to enhance heat transfer, and they often increase the rate of heat transfer from a surface several-fold. The car radiator is shown in Fig. below is an example of a finned surface. The closely packed thin metal sheets attached to the hot water tubes increase the surface area for convection and thus the rate of convection heat transfer from the tubes to the air many times. There are a variety of innovative fin designs available in the market, and they seem to be limited only by imagination

In the analysis of fins, we consider steady operation with no heat generation in the fin, and we assume the thermal conductivity k of the material to remain constant.

We also assume the convection heat transfer coefficient h to be constant and uniform over the entire surface of the fin for convenience in the analysis. We recognize that the convection heat transfer coefficient h, in general, varies along the fin as well as its circumference, and its value at a point is a strong function of the fluid motion at that point.

The value of h is usually much lower at the fin base than it is at the fin tip because the fluid is surrounded by solid surfaces near the base, which seriously disrupt its motion to the point of **"suffocating"** it, while the fluid near the fin tip has little contact with a solid surface and thus encounters little resistance to flow.

To create a tractable equation for the heat transfer of a fin, many assumptions need to be made:

1. Steady state
2. Constant material properties (independent of temperature)
3. No internal heat generation
4. One-dimensional conduction
5. Uniform cross-sectional area
6. Uniform convection across the surface area

1.12 Fin Materials Properties:

Aluminum Fins: Provide the best thermal conductivity, Stainless Steel Fins: Fight high external corrosion, Copper Fins: Provide the best heat transfer.

1.13 Application of Fins:

1. Air-cooled I.C. engines
2. Refrigeration condenser tubes
3. Electric transformers
4. Reciprocating air compressors
5. Example of surfaces where fins are used
   Semiconductor devices
6. Automobile radiator
7. Cooling Of Electronic components
8. Dry-type cooling towers
9. They are also used in newer technology such as hydrogen fuel cells.

CHAPTER 2

LITERATURE REVIEW

2.1 Literature review

Sandhya Mirapalli and P.S. Kishore [1]

In this paper heat transfer analysis is done by placing rectangular and triangular fins.The analysis was carried out from 200oC to 600oC by varying the temperature of the cylinder surface, ranging from 6 cm to 14 cm. Input parameters such as input density, heat transfer coefficient, thermal conductivity and fin thickness determine the output parameters such as heat flow rate, heat flux per unit mass, efficiency and effect. Comparison with rectangular fins.

M. Sudheer [2]

In the paper, detailed work has been carried out to develop a finite element method to estimate the temperature distribution of steady-state heat transfer and thermal stress caused by temperature differences in silicon carbide (SiC) ceramic finned tubes. Transfer device. The results obtained are shown in a series of temperature and thermal stress distribution curves. The annular fins have a rectangular, trapezoidal and triangular profile and are suitable for various radius ratios. The results show that the radius ratio and fin profile are important parameters affecting the temperature and thermal stress distribution of the annular fin, using Ansys.

M. Jain [3]

The main purpose of the paper is to analyze the heat dissipation of the fins by changing the geometry of the fins. Parametric model of fins has been developed to predict transient thermal behavior. Later, the model is created by changing the geometry, such as rectangles, circles, triangles, and fins with extensions. The modeling software used is CREO Parametric 2.0. Analyze using ANSYS 14.5. The material currently used to make fins is usually aluminum alloy 204, which has a 4 thermal conductivity of 110-150 W / m-° C. We are using aluminum alloy 6061 material to analyze the fins,aluminum alloy 6061 has a higher thermal conductivity, about 160-170W/m^{O}C. After the material is determined, the third step is to increase the heat transfer rate of the system by changing geometric parameters such as cross-sectional area, parameters, length, thickness, and the like.

Prajesh Paul, Ram C Sharma et al. [4]

They done research work through ansys on various profiles of fins and analyzed through various alloys of aluminium including Aluminium Nitride, Cupronickel and Inconel MA754.All the properties of the alloys were analyzed in this paper.

Dharma Rao and others [5]

Numerically solved the problem of laminar natural convection heat transfer in fin array. They consider a vertical pedestal with horizontal fins and solve the heat conduction equations in the energy control equation and fins They validated their numerical results using experimental data obtained from the literature, and discussed the effects of parameters such as parameter temperature, fin height and fin spacing on heat transfer rate of fin arrays.

A.A. Dahghan [6]

In this paper, natural convection, conduction and radiant heat transfer combined in an open top upright chamber containing discrete heat sources have been digitally modelled . The surface emissivity has changed, and the effects on the flow field and the thermal field have been determined for different Rayleigh values, and the comparison of numerical results with experimental observations shows that the accurate prediction of the flow field and the thermal field depends largely on Radiative heat transfer in the case of logarithmic values.

Pardeep Singh, Harvinder lal et al [7]

Has done research work on various profiles of fins with extensions and without extensions using software and up to a certain length and they concluded that with extensions heat transfer rate is better as compared to no extensions.

Jing Ma [8]

In this paper, Spectral Element Method (SEM) was developed to address coupled conduction, convection and radiative heat transfer in 3 moving trapezoidal, convex parabolic and concave parabolic porous fins. In these irregular porous fins, uneven heat generation, heat transfer coefficient and surface emissivity vary with temperature.

M.P. Shah [9]

In this paper, the Ansys APDL software was used to numerically study copper, AA1100, AA2011, AA3105 and other materials. Transient and steady-state analysis of cylindrical fins under convection and specified base temperature conditions. Specify the length of the fin, the thickness of the base and the thickness of the end. Specify the thermal conductivity of the fin material.

R.S.K. Reddy and other [10]

The purpose of this paper is to examine the effect of the fin shape of the heat sink on thermal performance. Various types of perforations on pin fins for efficient heat transfer under constant heat flux conditions. Performing 3D modeling and analysis using CATIA and ANSYS by using forced convection of the fan to complete numerical simulation, 12.0. This will help identify new fin topologies with better heat transfer characteristics than traditional planar fins. The main goal is to increase the rate of heat transfer through the fin surface and reduce the material cost.

R. Gupta [11]

Objective of the paper is to implementation is to increase the heat dissipation rate by using an invisible working fluid, only the air. We know that by increasing the surface area, the heat dissipation rate can be increased, so it is very difficult to design such a large complex engine. The main purpose of using these heat sinks is to cool the engine cylinders with air. The main purpose of the project is to analyze thermal properties by changing the geometry of the cylindrical fin material.

A.Aziz [12]

The purpose of the work in this paper is to two fold. The first objective is to obtain a finite element solution for a two-dimensional triangular fin and to provide heat transfer data for a wide range of Biot numbers and length to substrate thickness so that the information can be used for prediction and design purposes.. The second purpose is to report additional data for the rectangular fins to complement the results of Lau and Tan.

Kumar,G. and Kamal,R.S.and Dwivedi,A.and Yadav,A.S.and Patel,H .[13]

In their article provided experimental investigation to predict the performance of heated triangular fin array within a vertically direction and its effects in brief.

Daund,V.S.and Palande D.D.[14]

Investigated in Rectangular fin the effect of fin spacing, fin height, fin length on the performance of heat dissipation from the fin arrays. It is found that convection heat transfer rate depends on fin height and fin length. For a given fin spacing, the convection heat transfer rate from fins increases with fin height and length.

Hossain,Md.and Raiyan,Md.and Sayeed, J.and Ahamed,U.[15]

Suggested that Fin performance can be varied under various circumstances like, length of fin profile, coefficient of thermal conductivity, ambient temperature. Different fin profile needs to be chosen for different purpose

Charan el.al. [16]

Analyzed extended surfaces, which are commonly used to enhance convection heat transfer coefficient in a wide range of engineering applications. The conception of introducing perforations on the lateral surface of fin is to enhance heat rate effectively. From the research, it is evident that tip temperature is minimum for aluminum triangularly perforated with three perforations in it and heat is maximum for triangularly perforated with three perforations of aluminum material. Therefore it is concluded that a three triangle laterally perforated aluminum is most suitable for the fin applications.

Rajesh et.al [17]

Analyzed the thermal properties by varying geometry, material (Cu, Al alloy 6082), distance between the fins and thickness of cylinder fins. The fin models are created by varying geometry circular and also by varying thickness of the fins of both geometries. The 3D modelling software Pro/Engineer & Unigraphics were used. Thermal analysis was done on cylinder fins to determine variation temperature distribution over time. The analysis was done using ANSYS. By doing thermal analysis on the engine cylinder fins it has been concluded that it is helpful to know the heat dissipation inside the cylinder.

Mogaji et.al. [18]

Performed numerical analysis of heat flow through fin of a rectangular profile surface with and without considering radiation heat loss. The effects of physical parameters which include: length, thickness, fin metal type and emissivity on the fin performance have been comparatively studied. It was observed that heat dissipation rate for the fin with thermal radiation was higher than those without thermal radiation independently of the fin type metal considered in the study. It was also observed that fin thermal performance for aluminum and copper materials is appreciable when compare to stainless steel material.

2.2 Scope of The Work from Literature Review

The gist of the above papers is the comparison of heat dissipation from a heat sink in two different external conditions of natural convection and forced convection. It is observed that the heat dissipation in forced convection occurred better than that in the natural convection. Thus, experiment would be performed and results and discussions would be concluded to justify the various materials that yields a better heat transfer in the case of natural convection .

CHAPTER 3

MODELING OF FINS USING CATIA SOFTWARE

This chapter gives basic idea about a CAD package called CATIA, its features and tools and step by step procedure to design the profile of a fin.

3.1 Introduction:

Initially CATIA name is an abbreviation for Computer Aided Three-dimensional Interactive Application .The French Dassault Systems is the parent company and IBM participates in the software and marketing, and Catia invades broad sectors, and has been explained in the previous post position of CATIA between 3d modeling software program CATIA was started in 1977 by French Aircraft Manufacturer Avions Marcel Dassault System.

CATIA is classified under the following software packages:

CAD (Computer Aided Design)
CAM (Computer Aided Manufacturing)
CAE (Computer Aided Engineering)

Version that most of the people works on it now is CATIA V5 or fifth version, which is a rewriting and the code of the fourth edition. For the fifth version, there are versions from I to 20, for example, CATIA V5 RI 7, it Means CATIA fifth edition version seventeenth, While years system was adoption in the sixth edition, for example,CATIA V6 2011 means CATIA sixth edition version of Year 2011.

3.2 CATIA Modules

3.2.1 Sketcher:

This module is responsible for the implementation of two-dimensional shapes, in preparation for make a three-dimensional commands on it.

3.2.2 Part Design:

This module responsible for converting two-dimensional graphics to three dimensional objects which is most famous in Catia and is closely linked with sketcher module. The part design Module it is considered from most important modules, that used by the designer to get the additional advantage from cad which is stereotaxic drawing or three-dimensional drawing.

3.2.1 SKETCHER

INTRODUCTION:

This module is responsible for the implementation of two-dimensional shapes, in preparation for make a three-dimensional commands on it.

3.2.1.1 Toolbars in sketcher:

Profile Toolbar:

The Profile toolbar contains 2D geometry commands. These geometries range from the very simple (point, rectangle, etc...) to the very complex (splines, conics, etc...). The Profile toolbar contains many sub-tool bars. Most of these sub-toolbars contains different options for creating the sane geometry. For example, you can create a simple line, a line defined by two tangent points, or a line that perpendicular to a surface.

Profile toolbar Reading from Eft to right, the Profile toolbar contain the following commands.

The profile toolbar is shown in Fig.3.2.1.1.

Fig 3.2.1.1 Profile toolbar

1) Profile:

This command allows you to create a continuous set of lines and arcs connected together.

2) Rectangle / Predefined Profile toolbar:

The default top command is rectangle. Stacked underneath are several different commands used to create predefined geometries.

3) Circle / Circle toolbar

The default top command is circle. Stacked underneath are several different options for creating circles and arcs.

4) Spline / Spline toolbar:

The default top command is spline which is a curved line created by connecting a series of points.

5) Ellipse / Conic toolbar:

The default top command is ellipse. Stacked underneath are commands to create different conic shapes such as a hyperbola.

6) Line / line toolbar:

The default top command is line. Stacked underneath are several different options for creating lines.

7) Axis:

An axis is used in conjunction with commands hike mirror and shaft (revolve). It defines symmetry. It is a construction element so it does not become a physical part of your feature.

8) Point / Point toolbar:

The default top command is point. Stacked underneath are several different options for creating points.

9) Rectangle:

The Rectangle is defined by two corner points. The Sides of the rectangle are always horizontal and vertical.

10) Oriented Rectangle:

The rectangle is defined by three comer points. This allows you to create a rectangle whose sides are at an angle to the horizontal.

11) Parallelogram:

The parallelogram is defined by three comer points.

12) Elongated Hole:

The elongated hole or slot is defined by two points and a radius.

13) Cylindrical Elongated Hole:

The cylindrical elongated hob is defined by a cylindrical radius, two points and a radius.

14) Keyhole Profile:

Keyhole profile defined by two center points and two radii.

15) Hexagon:

The hexagon is defined by a center point and the radius of an inscribed circle.

16) Centered Rectangle:

The centered rectangle is defined by a center point and a comer point.

17) Centered Parallelogram

The centered parallelogram is defined by a center point (defined by two intersecting lines) and a comer point. Creates text that may be used to create a solid.

18) Circle:

A circle is defined by a center point and a radius.

19) Three Point Circle:

The three-point circle command allows you to create a circle using three circumferential points.

20) Circle Using Coordinates:

The circle using coordinates command allows you to create a circle by entering the coordinates for the center point and radius in a Circle Definition window.

21) Tri-Tangent Circle:

The tri-tangent circle command allows you to create a circle whose circumference is tangent to three chosen lines.

22) Three Point Arc:

The three-point arc command allows you to create an arc defined by three circumferential points.

23) Three Point Arc Starting with Limits:

The three-point arc starting with limits allows you to create an arc using a start, end, and midpoint.

24) Arc:

The arc command allows you to create an arc defined by a center point, and a circumference start and end point.

25) spline:

A spline is a curved profile defined by three or more points. The tangency and curvature radius at each point may be specified.

26) Connect:

The connect command connects two points or profiles with a spline.

27) Conic toolbar:

Reading from left to right, the Conic toolbar contains the following command.

28) Ellipse:

The ellipse is defined by a center point and major and minor axis points.

29) Parabola by Focus:

The parabola is defined by a focus, apex and start and end points.

30) Hyperbola by Focus:

The hyperbola is defined by a focus, center point, apex and start and end points.

31) Conic:

There are several different methods that can be used to create conic curves. 'Ihese methods give you a lot of flexibility when creating the above three types of curves.

32) Line:

A line is defined by two points.

33) Infinite Line:

Creates infinite lines that are horizontal, vertical or defined by two points.

34) Bisecting Line:

Creates an infinite line that bisects the angle created by two other lines.

35) Line to Curve:

The command allows you to create a line that starts anywhere and ends normal or perpendicular to another element.

36) Point by Clicking:

Creates a point by clicking the left mouse button.

37) Point by using coordinates:

Creates a point at a specified coordinate point.

38) Equidistant Points:

Creates equidistant points along a predefined path curve.

39) Intersection Point:

Creates a point at the intersection of two different elements.

Projection Point: projects a point of one element onto another.

Constraint toolbar:

Constraints can either be dimensional or geometrical. Dimensional constraints are used to constrain the length of an element, the radius or diameter of an arc or circle, and the distance or angle between elements. Geometrical constraints are used to constrain the orientation of one element relative to another. For example, two elements may be constrained to be perpendicular to each other. Other common geometrical constraints include parallel, tangent, coincident, concentric, etc... Reading from left to right:

The constraint toolbar is shown in Fig 3.2.1.2.

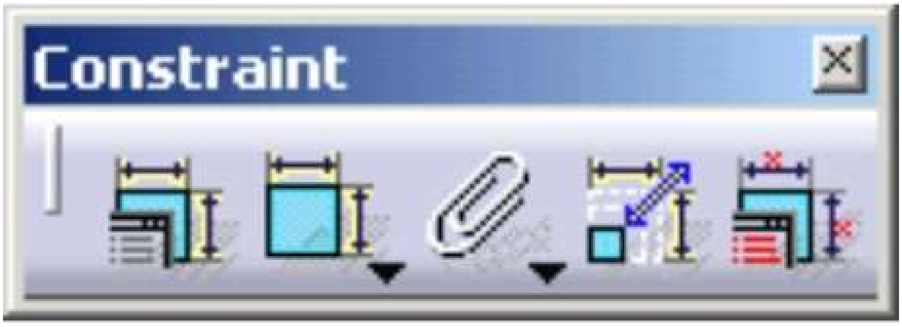


Fig 3.2.1.2 Constraint toolbar

1) Constraints Defined in Dialoged Box:

Creates geometrical and dimensional constraints between two elements.

2) Constraint:

Creates dimensional constraints. o Contact Constraint: Creates a contact constrain between two elements.

3) Fix Together: The fix together command groups individual entities together.

Auto Constraint: Automatically creates dimensional constraints.

4) Animate Constraint:

Animates a dimensional constrain between to limits.

5) Edit Multi-constraint:

The command allows you to edit all your sketch constraints in a single window.

3.2.2 Part Design:

INTRODUCTION:

The module is responsible for convening two-dimensional graphics to three dimensional objects which is most famous in Catia and is closely linked with sketcher module. The part design Module it is considered from most important modules, that used by the designer to get the additional advantage from cad programs which is stereotaxic drawing or three-dimensional drawing.

3.2.2.1 Toolbars in Part Design

Sketch-Based Features:

The toolbar is mainly used to create a solid feature from a 2D sketch/profile. Sketch based features toolbar is shown in Fig 3.2.2.1.

fig 3.2.2.1 Sketch based features toolbar

1) Pad:

The command is used to add material by extruding a sketch.

2) Pocket:

The command is used to remove material by extruding a sketch.

3) Shaft:

The command is used to add material by rotating a sketch.

4) Groove:

The command is used to remove material by rotating a sketch.

5) Rib:

The command is used to add material by sweeping a profile along a center curve.

6) Slot:

The command is used to remove material by sweeping profile along a center curve.

7) Multi-section1B Solid:

The command is used to add material by sweeping one or more planar section curves along one or more guide curves.

8) Removed Multi-sections Solid:

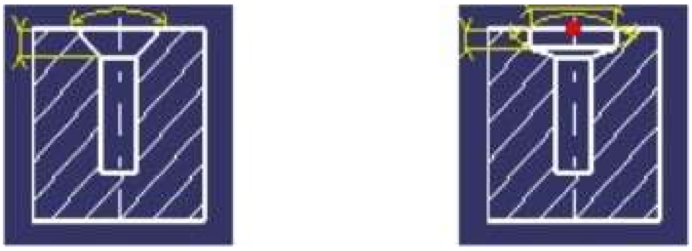
The command is used to remove material by sweeping one or more planar section curves along one or more guide curves.

9) Hole:

The command used for the circular material removal from the existing solid.

Various types of holes are shown in Fig 3.2.2.2:

Simple Tapered Counterbored



Countersunk Counter drilled

Fig 3.2.2.2 Various types of holes

3.3 DESIGN OF FINS

3.3.1 DESIGN OF RECTANGULAR FIN:

The design of Rectangular fin is shown in Fig 3.3.1.

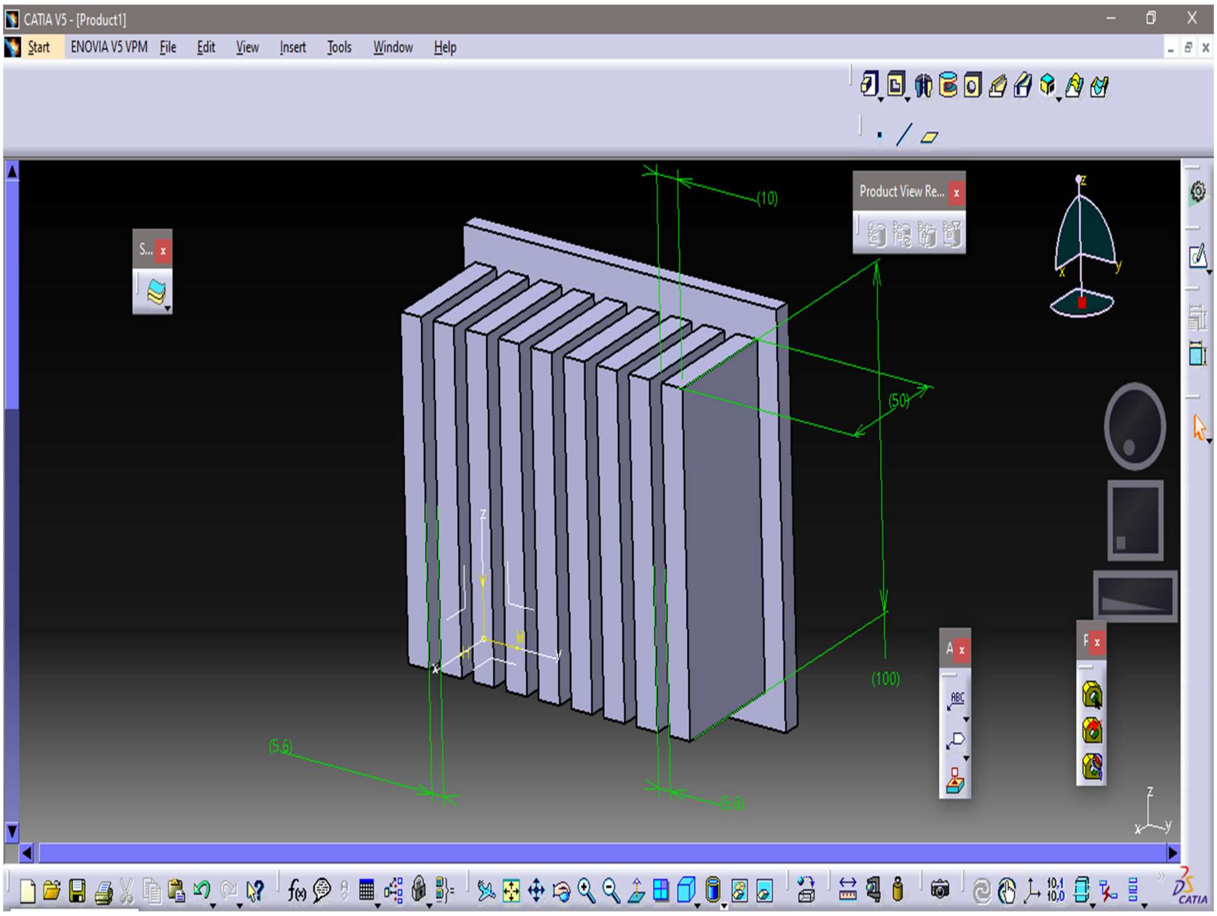


Fig 3.3.1 Design of Rectangular fin

CHAPTER 4

ANALYSIS OF FINS USING ANSYS

4.1 INTRODUCTION TO ANSYS

ANSYS was founded in 1970 by John Swanson. Swanson sold his interest in company to venture capitalists in 1993. ANSYS went public on NASDAQ in 1996. In the 2000s, ANSYS made numerous acquisitions of other engineering design companies, acquiring additional technology for fluid dynamics, electronics design, and other physics analysis.

4.1.1 Software

ANSYS develops and markets finite element analysis software used to simulate engineering problems. The software creates simulated computer models of structures, electronics, or machine components to simulate strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes. ANSYS is used to determine how a product will function with different specifications, without building test products or conducting crash tests. For example, ANSYS software may simulate how a bridge will hold up after years of traffic, how to best process salmon in a cannery to reduce waste, or how to design a slide that uses less material without sacrificing safety .

Most Ansys simulations are performed using the ANSYS Workbench software, which is one of the company's main products. Typically, ANSYS users break down larger structures into small components that are each modeled and tested individually. A user may start by defining the dimensions of an object, and then adding weight, pressure, temperature and other physical properties. Finally, the ANSYS software simulates and analyzes movement, Finally, the ANSYS software simulates and analyzes movement,fatigue, fractures, fluid flow, temperature distribution, electromagnetic efficiency, and other effects over time.

4.1.2 Basic program structure of ANSYS software

Treatment of engineering problems basically contains three main parts: create a model, solve the problem, analyses the results. ANSYS, like many other FE-programs, is also divided into three main parts (processors) which are called preprocessor, solution processer, post processer. Other software may contain only the preprocessing part or only the postprocessing part. During the analysis you will communicate with ANSYS via a Graphical User Interface (GUI), which is described below and seen in Figure 4.1.

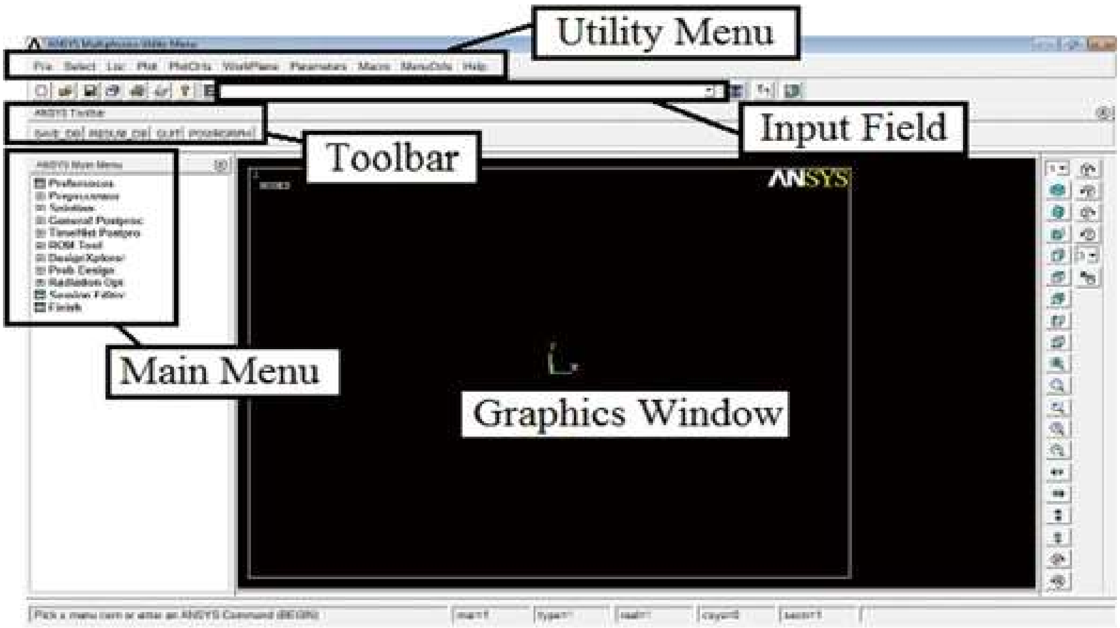


Fig 4.1 Graphical user Interface

Utility menu Here you can access and adjust properties about your session, such as file controls, listing and graphic controls.

Toolbar Push buttons to commonly used commands.

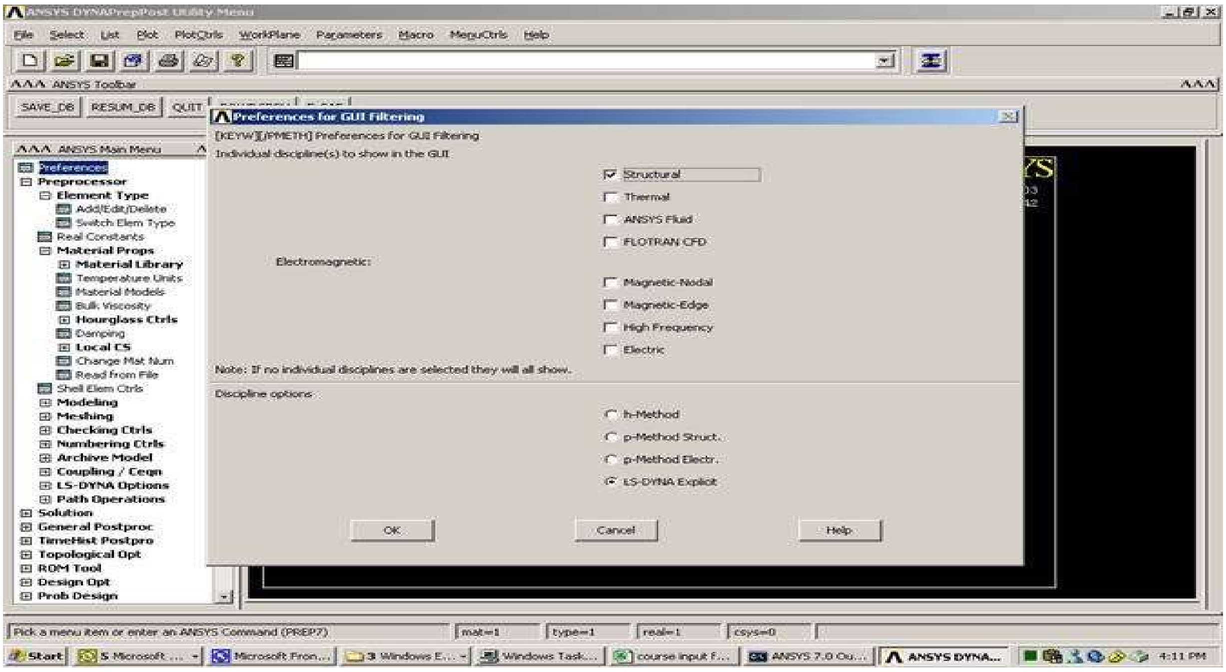
Main menu Here you can find the processors used when analyzing, your problem.

Graphics window in the graphics window your model is displayed: geometry, elements, visualization of results and so forth.

Input window You can type commands in the input window

4.1.3 Preferences

The starting step to define which analysis of a problem.The basic window is shown in Fig 4.1.1.



Preferences for GUI Filtering

[KEYW][/PMETH] Preferences for GUI Filtering

Individual discipline(s) to show in the GUI

☑ Structural

☐ Thermal

☐ ANSYS Fluid

☐ FLOTRAN CFD

Electromagnetic:

☐ Magnetic-Nodal

☐ Magnetic-Edge

☐ High Frequency

☐ Electric

Note: If no individual disciplines are selected they will all show.

Discipline options

○ h-Method

○ p-Method Struct.

○ p-Method Electr.

● LS-DYNA Explicit

Pick a menu item or enter an ANSYS Command (PREP7)

mat=1 type=1 real=1 csys=0

Fig 4.1.1 Basic window

4.1.4 Pre-processor

Within the preprocessor the model is set up. It includes a number of steps and usually in the following order:

Build geometry: Depending on whether the problem geometry is one, two or three dimensional, the geometry consists of creating lines, areas or volumes. These geometries can then, if necessary, be used to create other geometries by the use of Boolean operations. The key idea when building the geometry like this is to simplify the generation of the element mesh. Hence, this step is optional, but most often used. Nodes and elements can however be created from coordinates only.

Define materials: A material is defined by its material constants. Every element has to be assigned a particular material.

Generate element mesh: The problem is discretized with nodal points. The nodes are connected to form finite elements, which together form the material volume. Depending on the problem and the assumptions that are made, the element type has to be determined. Common element types are truss, beam, plate, shell and solid elements. Each element type may contain several subtypes, e.g. 2D 4-noded solid, 3D 20-noded solid elements. Therefore, care has to be taken when the element type is chosen.

The element mesh can in ANSYS be created in several ways. The most common way is that it is automatically created, however more or less controlled. For example, you can specify a certain number of elements in a specific area, or you can force the mesh generator to maintain a specific element size within an area. Certain element shapes or sizes are not recommended and if these limits are violated, a warning will be generated in ANSYS. It is up to the user to create a mesh which is able to generate results with a sufficient degree of accuracy.

4.2 ANSYS Work Bench

ANSYS Workbench is the framework upon which the industry’s broadest suite of advanced engineering simulation technology is built. An innovative project schematic view ties together the entire simulation process, guiding the user every step of the way. Even complex multi physics analyses can be performed with drag-and-drop simplicity.

The ANSYS Workbench platform automatically forms a connection to share the geometry for both the fluid and structural analysis, minimizing data storage and making it easy to study the effects of geometry changes on both Analysis. In addition, a connection is formed to automatically transfer pressure loads from the fluid analysis to the structural analysis.

The ANSYS Workbench interface is arranged into two primary areas: The toolbox and the project Schematic. The toolbox contains the system templates that you can use to build a project. The project Schematic is the area if the interface where you will manage your project. The new project schematic view shows an overall view of the entire simulation project. Engineering intent, data relationships and the state of the entire project are visible at a glance, even for complex analysis involving multiple physics. In addition to this, you will see a menu bar and a toolbar with frequently used functions. You can also use context menus, accessible via a right-mouse click, on schematic items and cells. Context menus provide capabilities to add to and modify projects. The entire process is persistent. Changes can be made to any portion of the analysis and the ANSYS Workbench platform will manage the execution of the required applications to update the project automatically, dramatically reducing the cost of performing design iterations.

ANSYS Workbench is a project-management tool. It can be considered as the top-level interface linking all our software tools. Workbench handles the passing of data between ANSYS Geometry/ Mesh/Post processing tools. This greatly helps project management. You do not worry about the individual files in disk (geometry, mesh etc.). Graphically, you can see at-a-glance how a project has been built. Because Workbench can manage the individual applications and pass data between them, it is easy to automatically perform design studies (parametric analyses) for design optimization.

4.2.1 ANSYS WORK BENCH FEATURES

- Bidirectional, parametric links with all major CAD systems.
- Integrated, analysis-focused geometry modelling, repair and simplification via ANSYS Design Modeler.
- Highly automated, physics-aware meshing.
- Automatic contact detection.
- Unequalled depth of capabilities within individual physics disciplines.
- Unparalleled breadth of simulation technologies.
- Complete analysis systems that guide the user start-to-finish through an analysis.
- Comprehensive multi physics simulation with drag-and-drop ease of use.
- Flexible components enable tools to be deployed to best suit engineering intent.
- Innovative project schematic view allows engineering intent, data relationships, and the state of the project to be comprehended at a glance.

4.3 Introduction to Steady State Thermal Analysis

4.3.1 Definition of steady state analysis

The ANSYS/Multiphysics, ANSYS/Mechanical, ANSYS/FLOTRAN, and ANSYS/Thermal products support steady-state thermal analysis. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineer/analysts often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. A steady state analysis also can be the last step of a transient thermal analysis, performed after all transient effects have diminished.

You can use steady-state thermal analysis to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convections
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries.

A steady state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material do vary with temperature, so the analysis usually is nonlinear. Including radiation effects also makes the analysis nonlinear.

4.3.2 Available elements for thermal analysis

1. Two-Dimensional Solid Elements
2. Three-Dimensional Solid Elements
3. Radiation Link Elements
4. Conducting Bar Elements
5. Convection Link Elements
6. Shell Elements
7. Coupled-Field Elements
8. Specialty Elements

4.3.3 Tasks in a thermal analysis

The procedure for doing a thermal analysis involves three main tasks:

- Build the model.
- Apply loads and obtain the solution and review the result

4.3.3.1 Creating model geometry/Build the model

There is no single procedure for building model geometry; the tasks you must perform to create it vary greatly, depending on the size and shape of the structure you wish to model. Therefore, the next few paragraphs provide only a generic overview of the tasks typically required to build model geometry.

The first step in creating geometry is to build a solid model of the item you are analyzing. You can use either predefined geometric shapes such as circles and rectangles (known within ANSYS as primitives), or you can manually define nodes and elements for your model. Two-dimensional primitives are called areas, and 3-D primitives are called volumes.

Model dimensions are based on a global coordinate system. By default, the global coordinate system is Cartesian, with X, Y, and Z axes; however, you can choose a different coordinate system if you wish. Modelling also uses a working plane-a movable reference plane used to locate and orient modelling entities. You can turn on the working plane grid to serve as a "drawing tablet" for your model.

You can tie together, or sculpt, the modelling entities you create via Boolean operations, For example, you can add two areas together to create a new, single area that includes all parts of the original areas. Similarly, you can overlay an area with a second area, then subtract the second area from the first; doing so creates a new, single area with the overlapping portion of area.

Once you finish building your solid model, you use meshing to "fill" the model with nodes and elements.

4.3.3.2 Applying loads and obtaining the solution

You must define the analysis type and options, apply loads to the model, specify load step options, and initiate the finite element solution.

Applying Loads

You can apply loads either on the solid model (key points, lines, and areas) or on the finite element model (nodes and elements). You can specify loads using the conventional method of applying a single load individually to the appropriate entity, or you can apply complex boundary conditions via TABLE type array parameters.

You can apply five types of thermal loads:

1)Constant Temperatures (TEMP)

These are DOF constraints usually specified at model boundaries to imposea known, fixed temperature.

2) Heat Flow Rate (HEAT)

These are concentrated nodal loads. Use them mainly in line-element models (conducting bars, convection links, etc.) where you cannot specify convections and heat fluxes. A positive value of heat flow rate indicates heat flowing into the node (that is, the element gains heat). If both TEMP and HEAT are specified at a node, the temperature constraint prevails.

Note**:** If you use nodal heat flow rate for solid elements, you should refine the mesh around the point where you apply the heat flow rate as a load, especially if the elements containing the node where the load is applied have widely different thermal conductivities otherwise you may get an unphysical range of temperature. Whenever possible, use the alternative option of using the heat generation rate load or the heat flux rate load. These options are more accurate, even for a reasonably coarse mesh.

3) Convections (CONV)

Convections are surface loads applied on exterior surfaces of the model to account for heat lost to (or gained from) a surrounding fluid medium. They are available only for solids and shells. In line-element models, you can specify convections through the convection link element (LINK34), or the thermal-fluid link element (FLUID66).

4) Heat Fluxes (HEAT)

Heat fluxes are also surface loads. Use them when the amount of heat transfer across a surface (heat flow rate per area) is known or is calculated through a FLOTRAN CFD analysis. A positive value of heat flux indicates heat flowing into the element. Heat flux is used only with solids and shells. An element face may have either CONV or HFLUX (but not both) specified as a surface load. If you specify both on the same element face, ANSYS uses what was specified *last*.

4.4 Analytical Modelling

4.4.1 Heat transfer principle

Heat sink works on the principle of Fourier's law of heat conduction, simplified to a one-dimensional form in the X-direction, shows that when there is a temperature gradient in a body, heat will be transferred from the higher temperature region to the lower temperature region. The rate at which heat is transferred by conduction, is proportional to the product of the temperature gradient and the cross-sectional area through which heat is transferred

$$Q_i = -kA \times (dT/dx) \qquad (4.1)$$

Consider a heat sink in a duct, where air flows through the duct. It is assumed that the heat sink base is higher in temperature than the air. Applying the conservation of energy, for steady-state conditions, and Newton's law of cooling to the temperature nodes shown in the diagram gives the following set of equations:

$$Q=mc_{p,in}(T_{air,out}-T_{air,in}) \qquad (4.2)$$

Where, $$T_{air,av}=(T_{air,in}+T_{air,out})/2 \qquad (4.3)$$

4.4.2 Assumptions for thermal analysis of heat Fin:

The performance of the heat sink is determined by the various parameters like the heat transfer coefficient, Fin efficiency, etc.

Heat Transfer Coefficient (h)

To define the heat transfer coefficient, consider the Newton's law of convection given by the equation

$$Q=hA_s(T_s-T_a) \qquad h=Q/\,A_s(T_s-T_a) \tag{4.4}$$

Where,

h = heat transfer coefficient of fin,

P = Perimeter of fin = 2(B+D),

L = Length of pin fin.

k = thermal conductivity of Aluminum fin = 200 W/mK

A= Area of fin = B x D

T_a = Ambient temperature = 25 ^{0}C

T_s = Surface temperature = 200 ^{0}C

This analysis is based on the following assumptions:

1. The fins are with adiabatic tip.
2. The fluid, air is assumed to be incompressible throughout the process.
3. Air properties are taken at film temperature.
4. There are no heat sources within the fin itself.
5. The radiation heat transfer is negligible.
6. The temperature at the base of the fin is uniform.

4.4.2 ENGINEERING DATA:

The engineering data of Aluminum alloy 6063 and Aluminum alloy 7068 are shown in Table 3 and Table 4 respectively.

S. NO	PROPERTIES	VALUES
1.	DENSITY	2700 Kg/m^3
2.	THERMAL CONDUCTVITY	200 W/m K
3.	SPECIFIC HEAT	900 J/Kg K

TABLE 3. PROPERTIES OF ALUMINIUM ALLOY 6063

S. NO	PROPERTIES	VALUES
1.	DENSITY	2800 Kg/m^3
2.	THERMAL CONDUCTVITY	190 W/m K
3.	SPECIFIC HEAT	963 J/Kg K

TABLE 4. PROPERTIES OF ALUMINIUM ALLOY 7068

4.4.4 CALCULATIONS:

T_s=200 °C, T_a=25 °C

$T=(T_s+T_a) / 2$

$= (200+25)/2$

$= 112.5$ °C

At 112.5 °C, Properties of dry air

$v=24.584 \times 10^{-6}$ m²/s

k=0.0329 W/m K

Pr=0.68675

$\beta=1/T$

$=1 / (112.5+273)$

$=0.0026$

$Gr = (g\beta(T_s-T_a)L^3) / v^2$ (where Gr grashofs number)

$Gr= 9.81 \times 0.0026 \times (200-25) \times (0.1)^3 / (24.58 \times 10^{\wedge}-6)^2$

Gr= 7380294.89

Ra= Gr x Pr= 7380294.89 x 0.68675 = 5274442.5157

$$S_{opt}\,[14] = 2.714 \times \frac{L}{(Ra)^{1/4}}$$

$=2.714 \times 0.1/ (5274442.5157)^{0.25}$

= 0.00566 m

$$h = 1.31 \times k / s_{opt}$$

Where

S_{opt} = Optimal spacing between fins

$$h = 1.31 \times 0.0329/0.00566$$

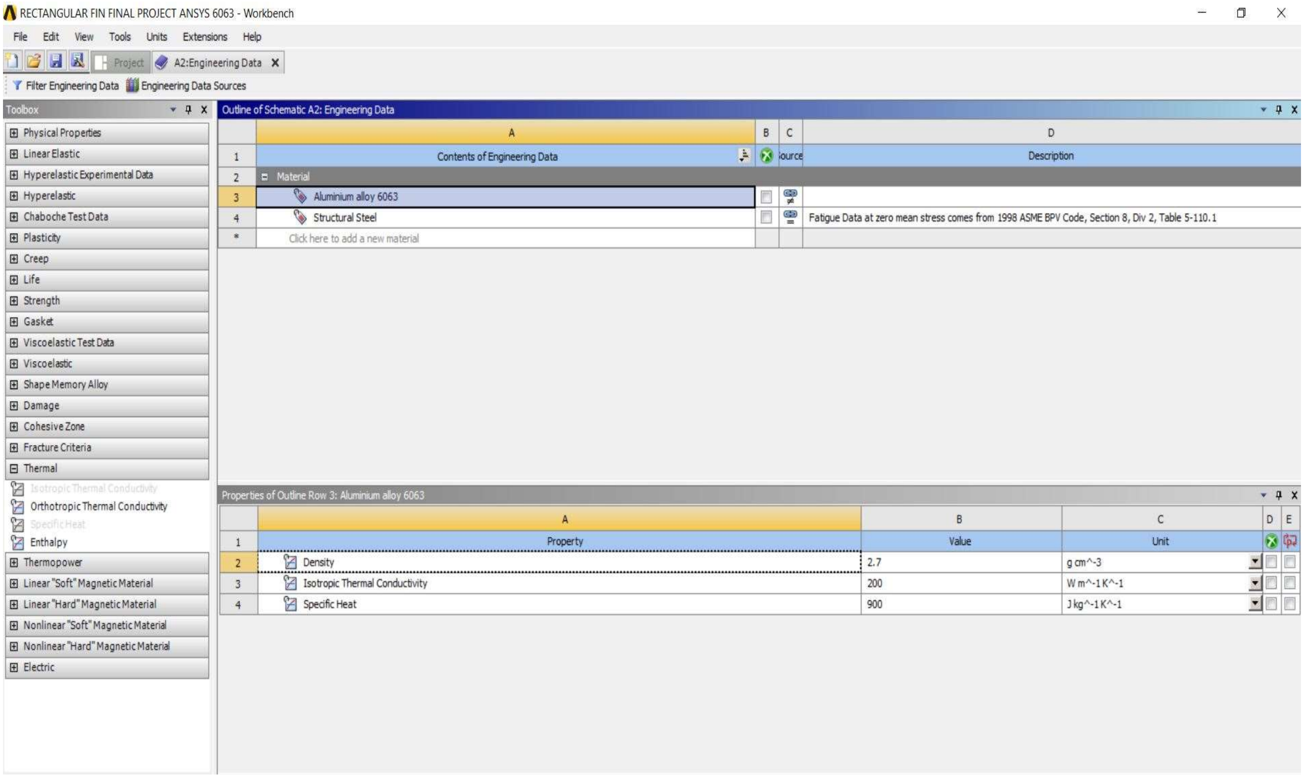
$$h = 7.614\ w/m^2\ {}^{o}C$$

4.5 STEADY STATE THERMAL ANALYSIS ON RECTANGULAR FIN

The 3d assembly model designed in solid work is saved as IGES format to do the analysis in ANSYS work bench. The work bench extracts the information of dimensions of the model and open to select the type of analysis that is required .

The engineering data of aluminum alloy 6063 is added to component with thermal conductivity of 200 W/mk, specific heat of 900 J/kg K, density of 2.7 g/cm^3.

Material selection For AA 6063 in Ansys workbench is shown in Fig 4.5.1.



Outline of Schematic A2: Engineering Data

	A	B	C	D
1	Contents of Engineering Data		ource	Description
2	Material			
3	Aluminium alloy 6063	☐		
4	Structural Steel	☐		Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material			

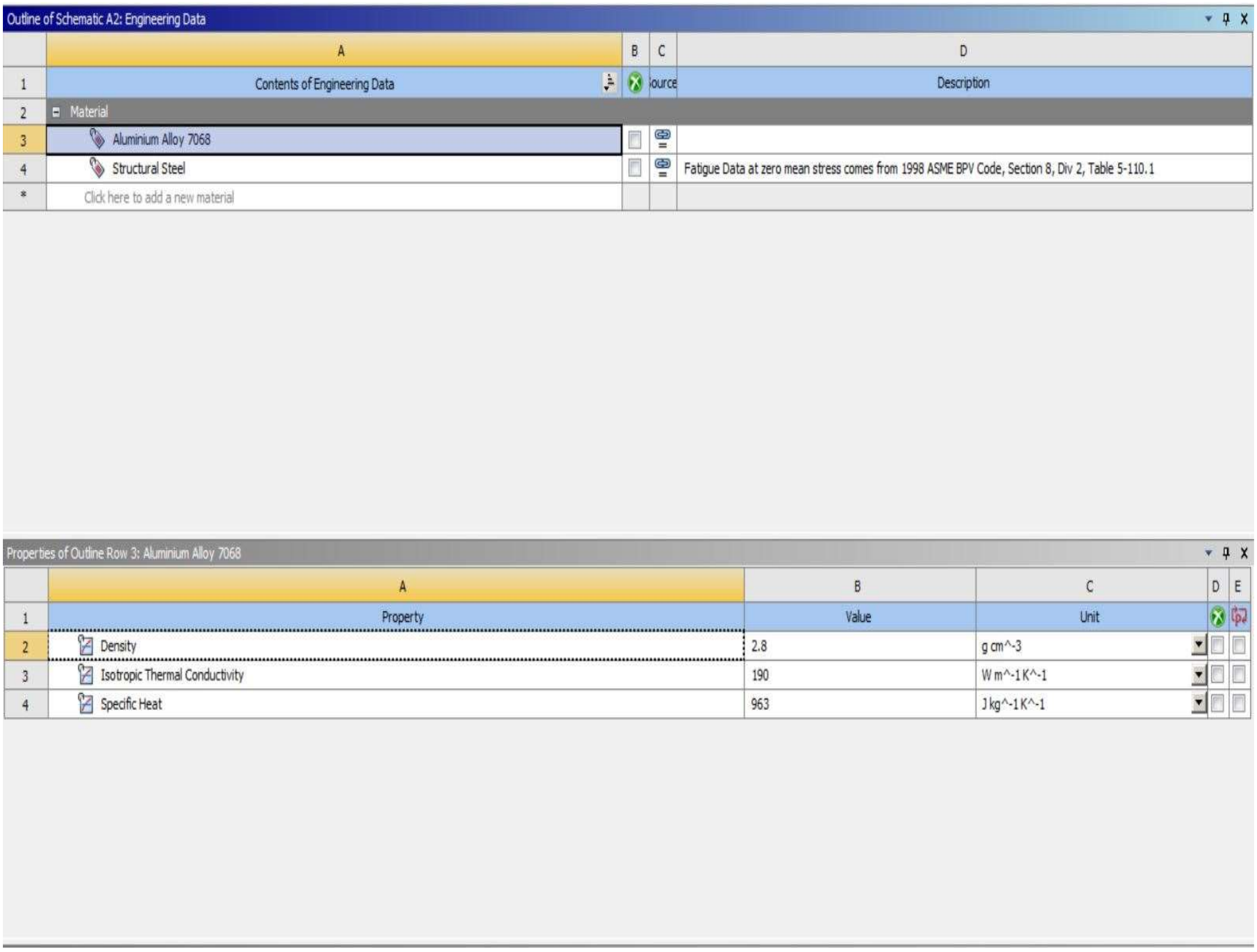
Properties of Outline Row 3: Aluminium alloy 6063

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	2.7	g cm^-3	☐	☐
3	Isotropic Thermal Conductivity	200	W m^-1 K^-1	☐	☐
4	Specific Heat	900	J kg^-1 K^-1	☐	☐

Fig 4.5.1 Material selection For AA 6063 in Ansys workbench

In the same way, the engineering data of aluminum alloy 7068 is added to component with thermal conductivity of 190 W/mk, specific heat of 963 J/kg K, density of 2.8 g/cm^3.

Material selection for AA 7068 in Ansys workbench is shown Fig 4.5.2.



Outline of Schematic A2: Engineering Data

	A	B	C	D
1	Contents of Engineering Data		ource	Description
2	Material			
3	Aluminium Alloy 7068	☐		
4	Structural Steel	☐		Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material			

Properties of Outline Row 3: Aluminium Alloy 7068

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	2.8	g cm^-3	☐	☐
3	Isotropic Thermal Conductivity	190	W m^-1 K^-1	☐	☐
4	Specific Heat	963	J kg^-1 K^-1	☐	☐

Fig 4.5.2 Material selection for AA 7068 in Ansys workbench

4.6 MESH GENERATION

The fin model is imported in to the work bench design modeler and meshed with a four node three dimensional tetrahedron element. The meshed model of the fin is shown in the Fig 4.6.1 and the mechanical Ansys workbench is used to mesh the fin.

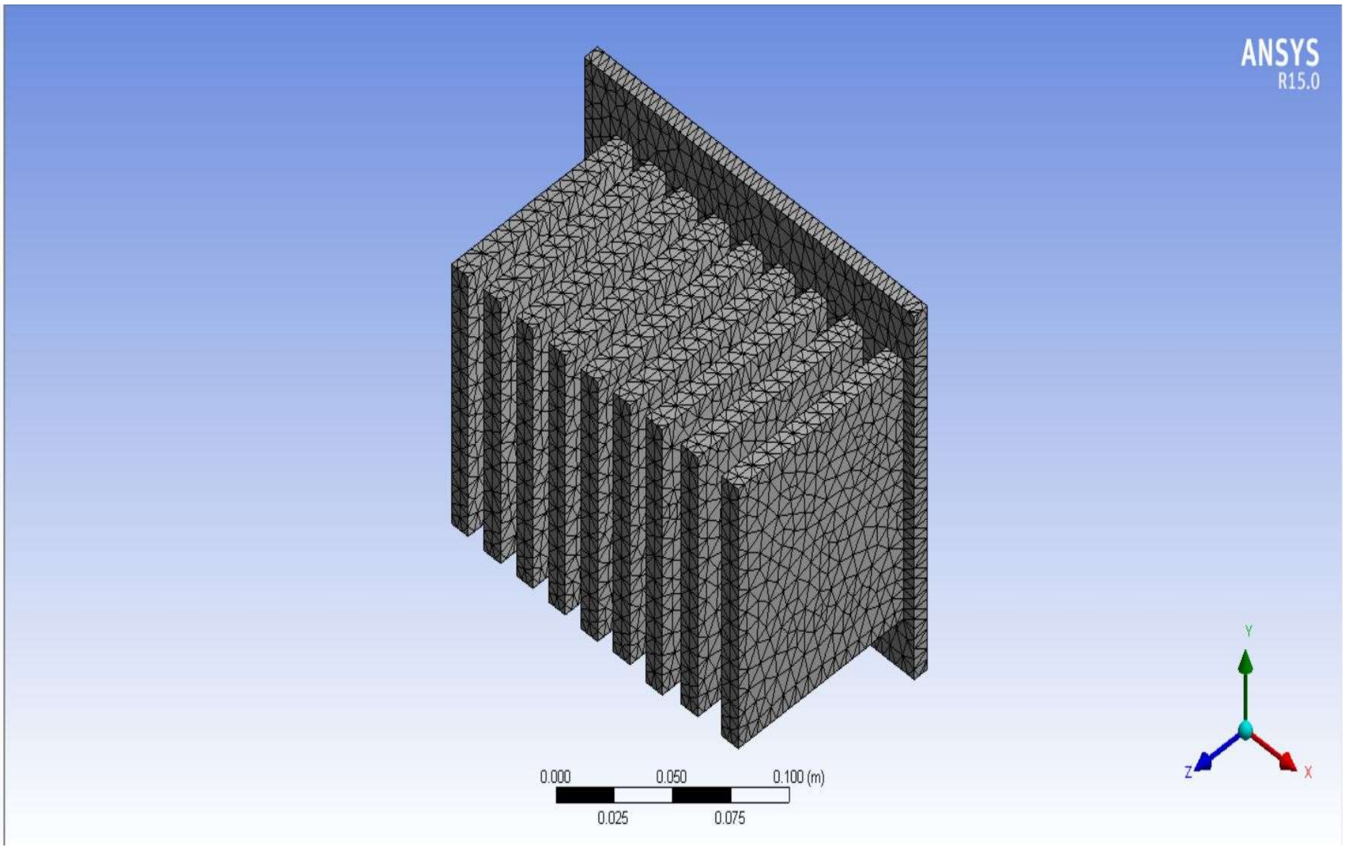


Fig 4.6.1 Meshing of fin model

4.7 APPLYING CONVECTION

Convection is applied for aluminum alloy 6063 and 7068. The value of convection coefficient is 7.614 w/m$^{2\circ}$c at a temperature of 25°c.

Isometric view of applied Convection is shown in Fig 4.7.1.

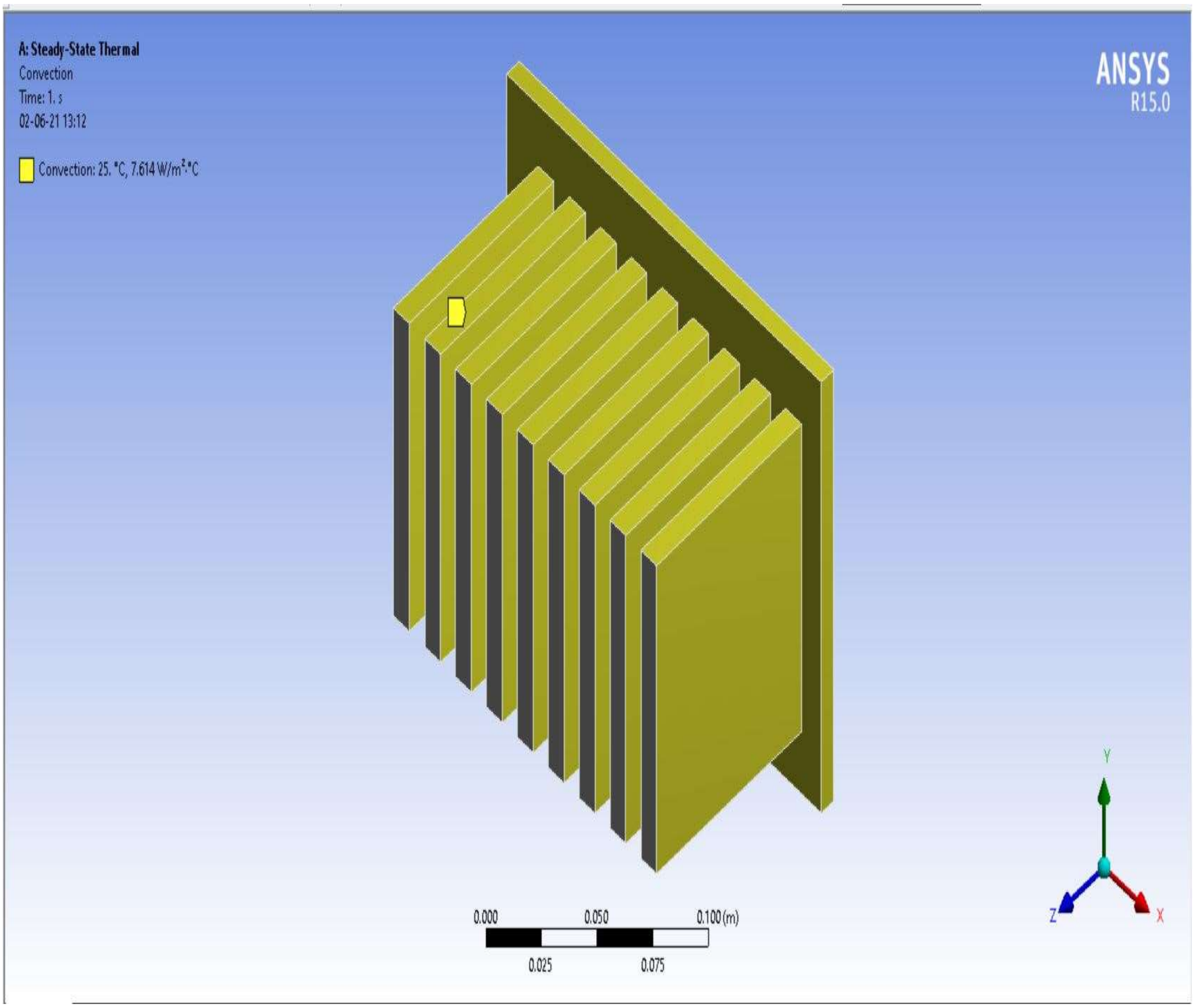


Fig 4.7.1 Isometric view of applied Convection

CHAPTER 5

RESULTS AND DISCUSSIONS

Steady-steady thermal analysis is evaluating the thermal equilibrium of a system in which the temperature remains constant over a time. In other words, steady state thermal analysis involves assessing the equilibrium state of a system subject to constant heat loads and environment conditions.

For a steady state thermal analysis, an amount of heat flow is given as input to the base plate of each fin considered in the analysis. A constant convection heat transfer coefficient of 7.614 W/m^2 ^oc and ambient temperature 25^0 C have been applied to the all remaining surfaces of a fin considered in the analysis. Using Ansys workbench, the maximum and minimum temperature, maximum and minimum heat flux of a fin have been estimated.

Results obtained in temperature distribution of aluminum alloy 6063 and 7068 are shown below. And also results obtained in heat flux of alumium alloy 6063 and 7068 are shown below.

5.1 Temperature Distribution

5.1.1 Aluminum alloy 6063

Rectangular fin

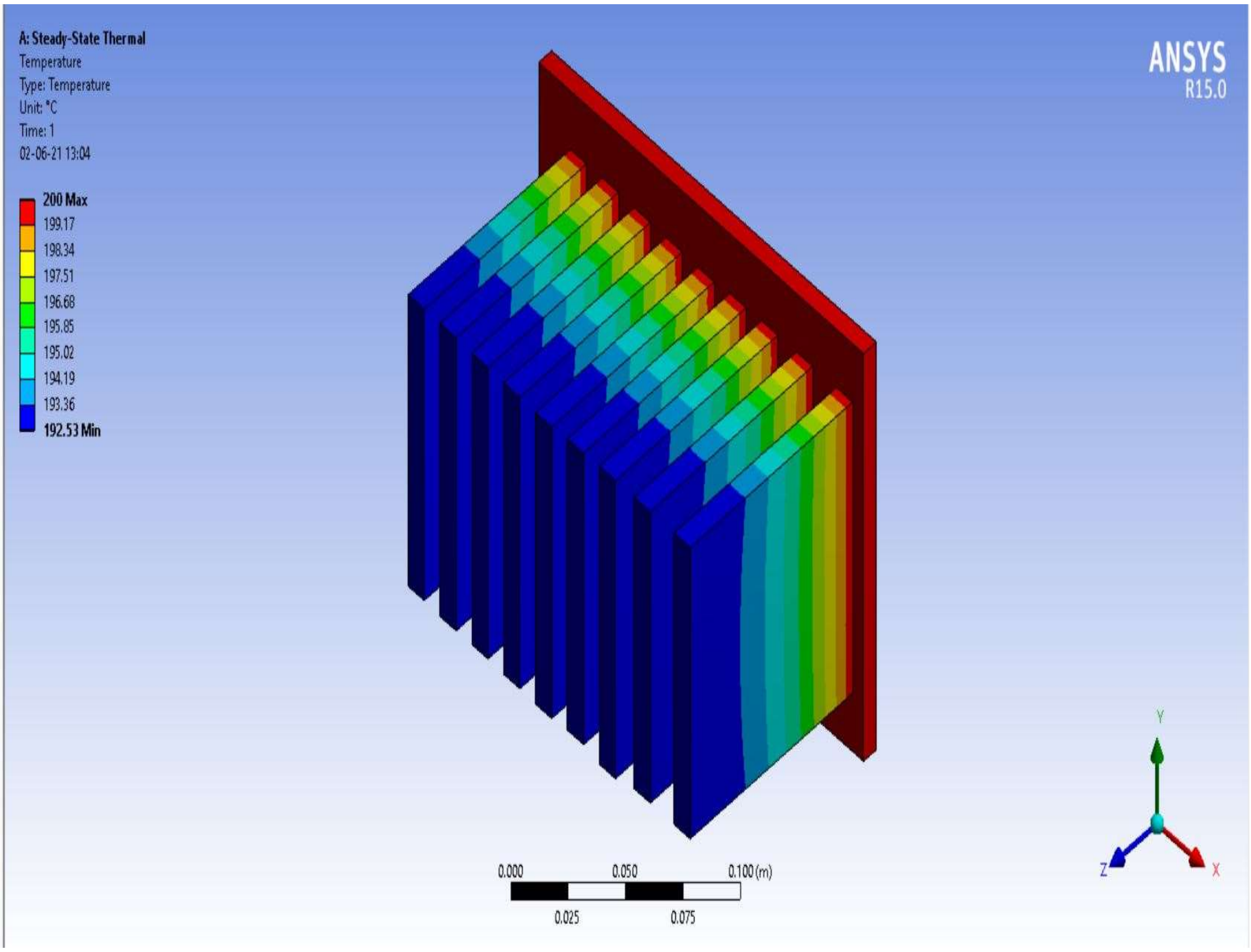
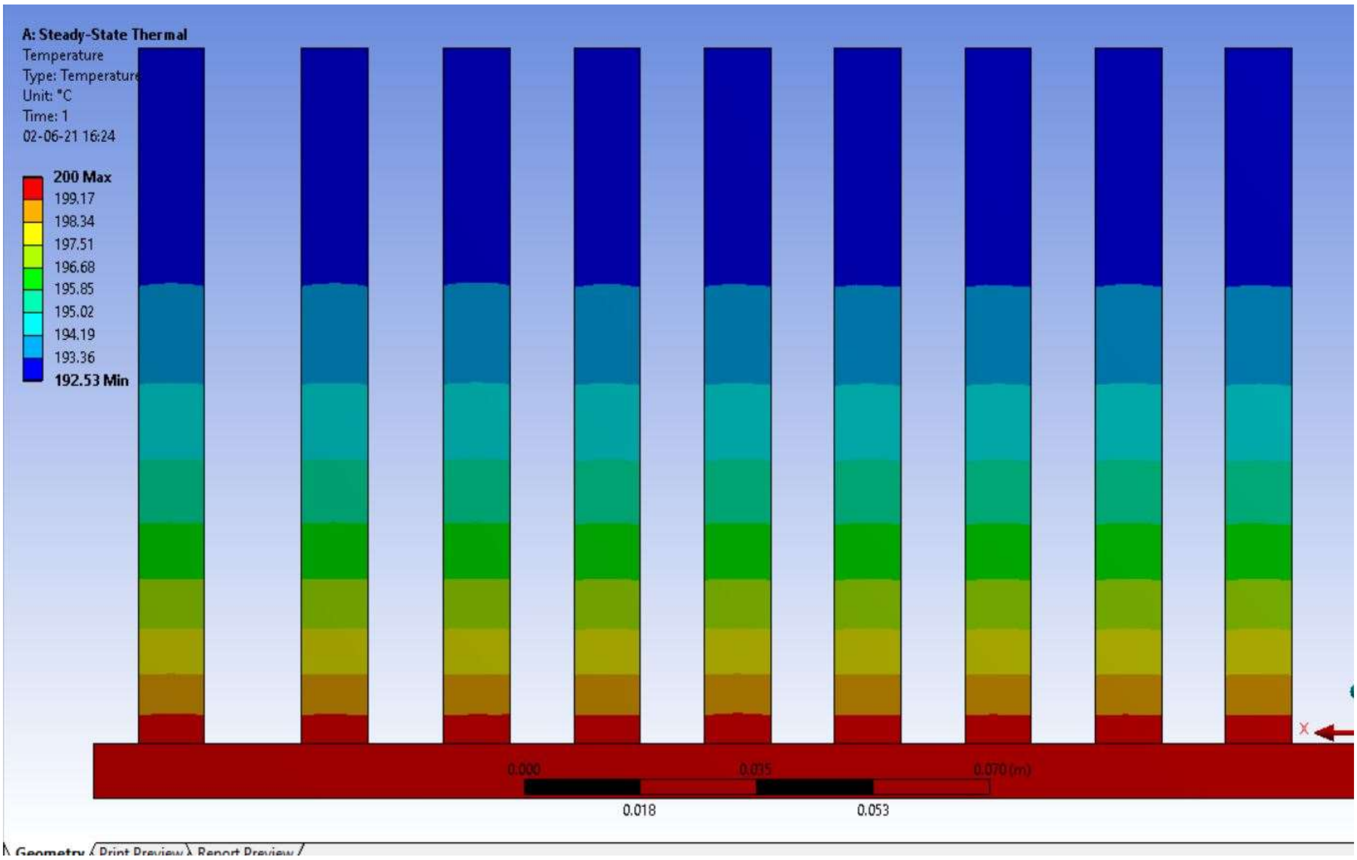


Fig 5.1.1(a) Temperature distribution of Rectangular Fin (Isometric view)

The temperature distribution for aluminum alloy 6063 is obtained with 200°C of maximum temperature at the center of base fin and minimum temperature of 192.53 °C is obtained at the tip of the fins. The simulated is shown in Fig 5.1.1(a) and Fig 5.1.1(b).



A: Steady-State Thermal
Temperature
Type: Temperature
Unit: °C
Time: 1
02-06-21 16:24

Fig 5.1.1(b) Temperature distribution of Rectangular Fin (Frontview)

5.1.2 Aluminum alloy 7068

Rectangular fin

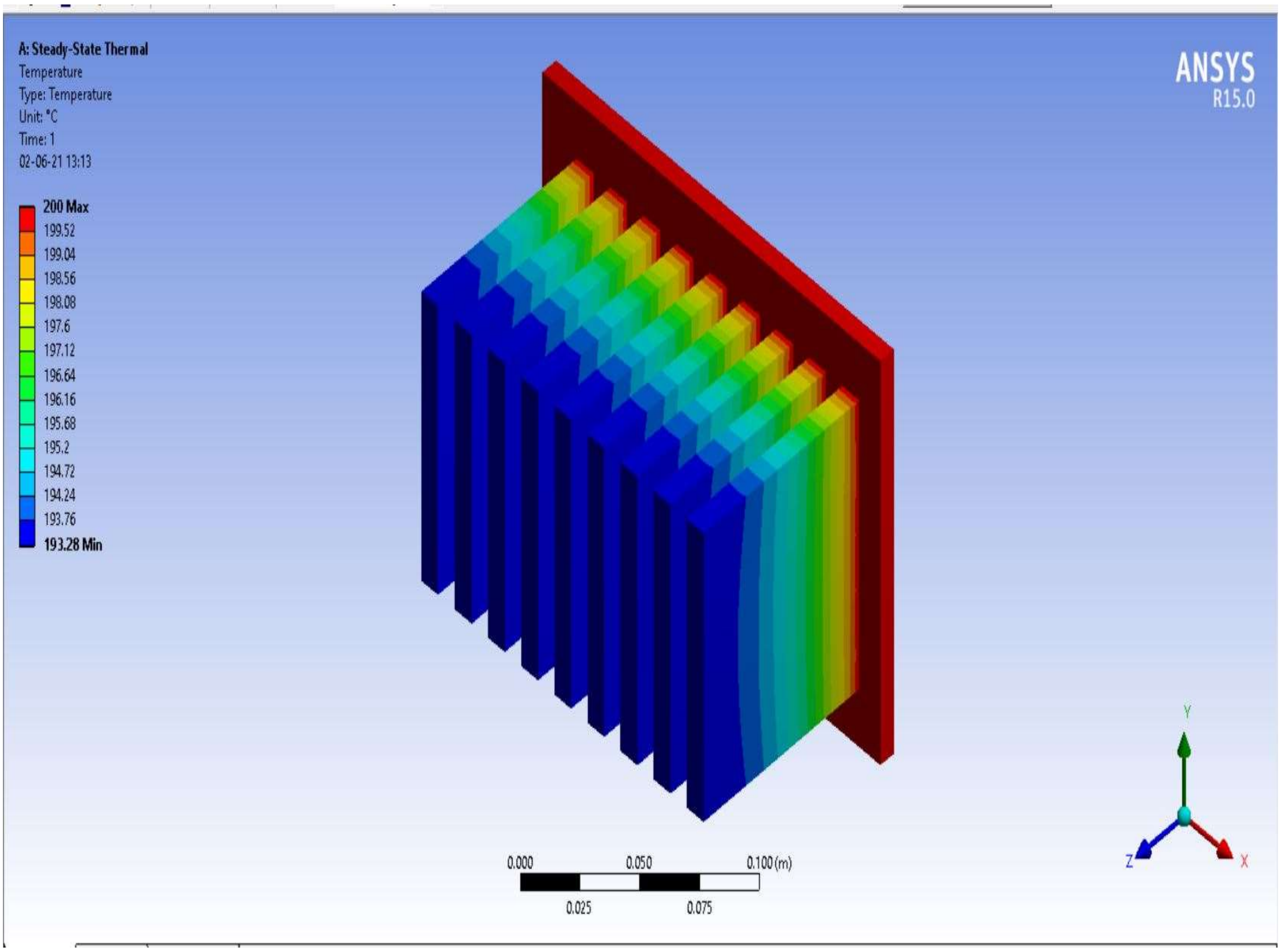


Fig 5.1.2(a) Temperature Distribution of Rectangular Fin (Isometric view)

The temperature distribution for aluminum alloy 6063 is obtained with 200°C of maximum temperature at the center of base fin and minimum temperature of 193.28 °C is obtained at the tip of the fins. The simulated is shown in Fig 5.1.2(a) and Fig 5.1.2(b) .

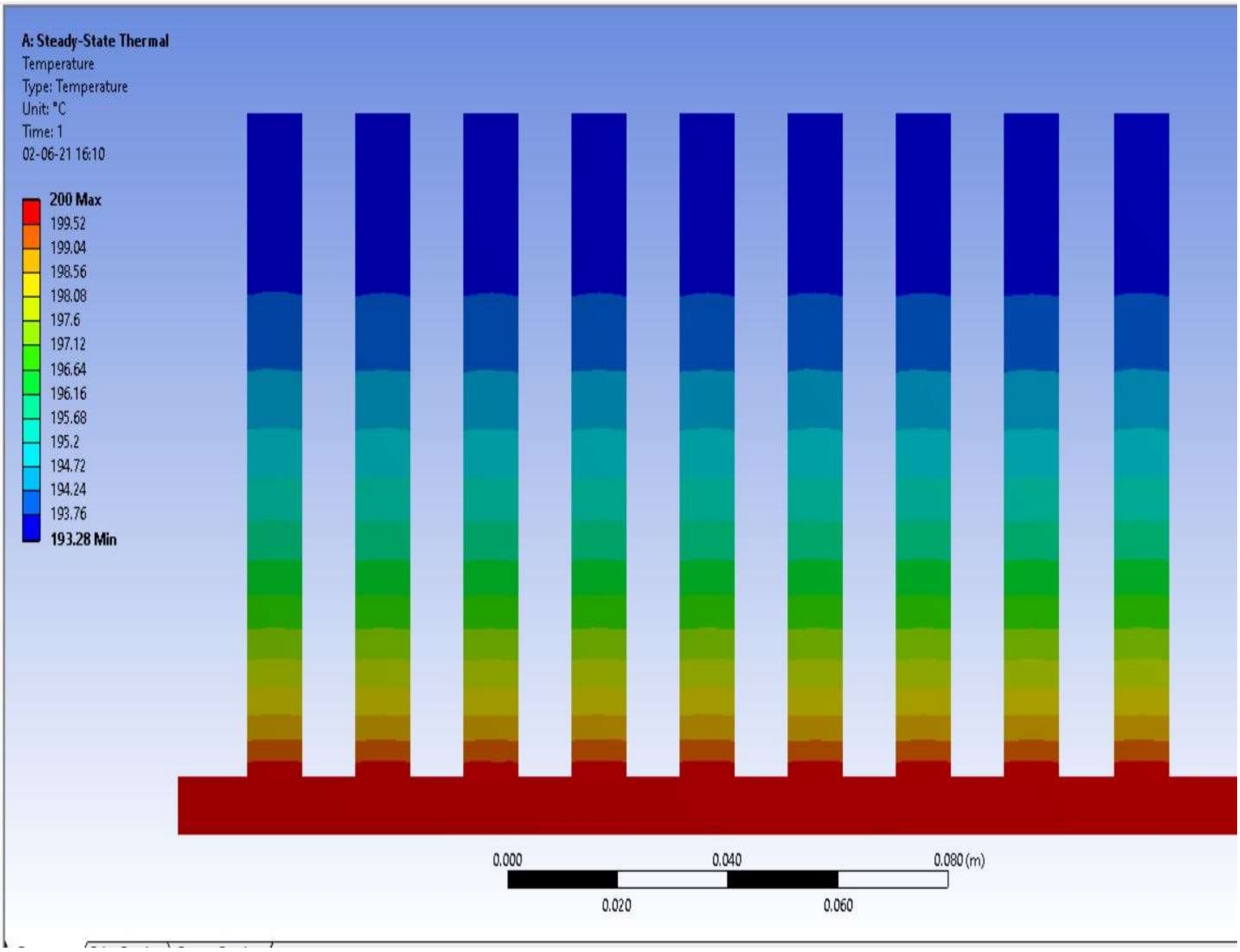


Fig 5.1.2(b) Temperature Distribution of Rectangular Fin (Front view)

5.2 Heat Flux Distribution

5.2.1 Aluminum alloy 6063

Rectangular fin

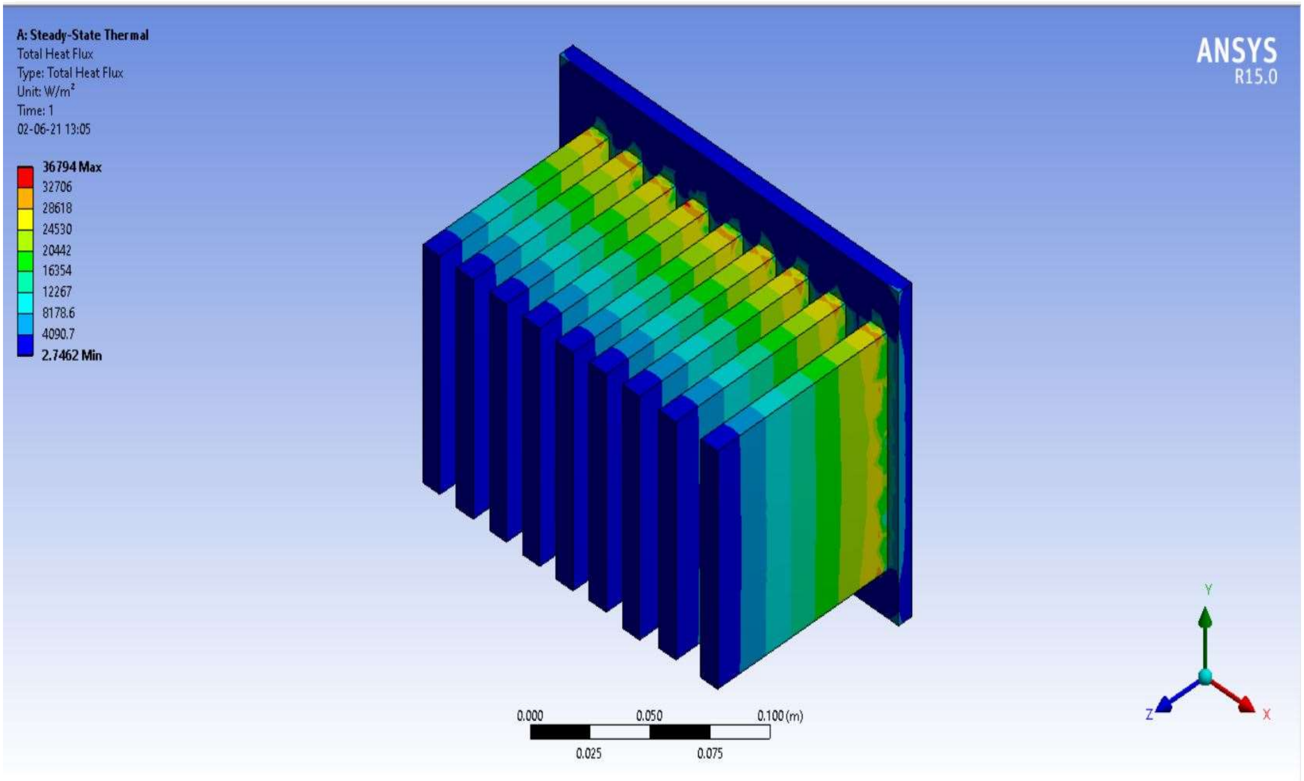


Fig 5.2.1(a) Heat flux Distribution of Rectangular Fin

(Isometric view)

The total heat flux distribution of orientation to Z-axis is obtained with a maximum of 36794 W/m^2 and minimum of 2.7462 W/m^2. The simulated result is as shown in Fig 5.2.1(a) and Fig 5.2.1(b).

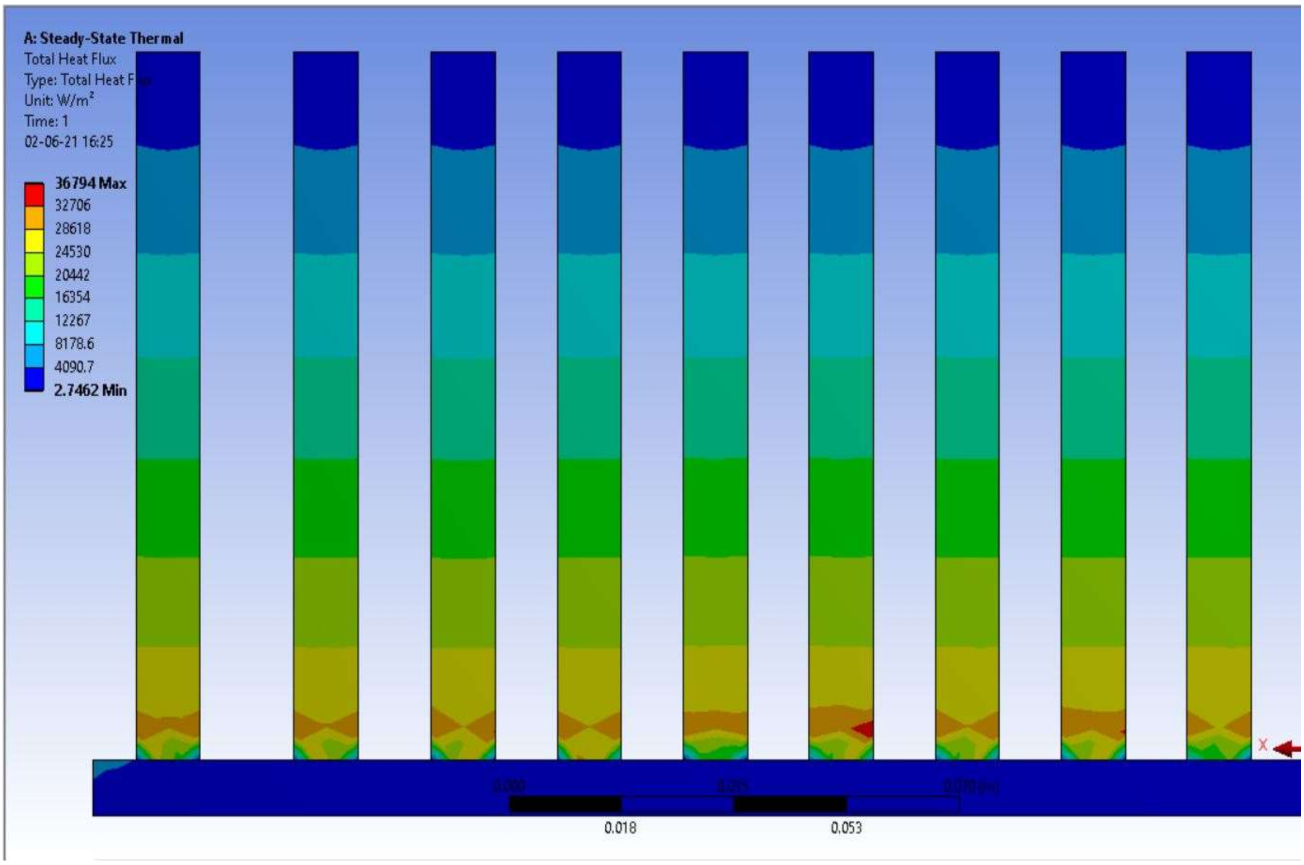


Fig 5.2.1(b) Heat flux Distribution of Rectangular Fin

(Front view view)

5.2.1 Aluminum alloy 7068

Rectangular fin

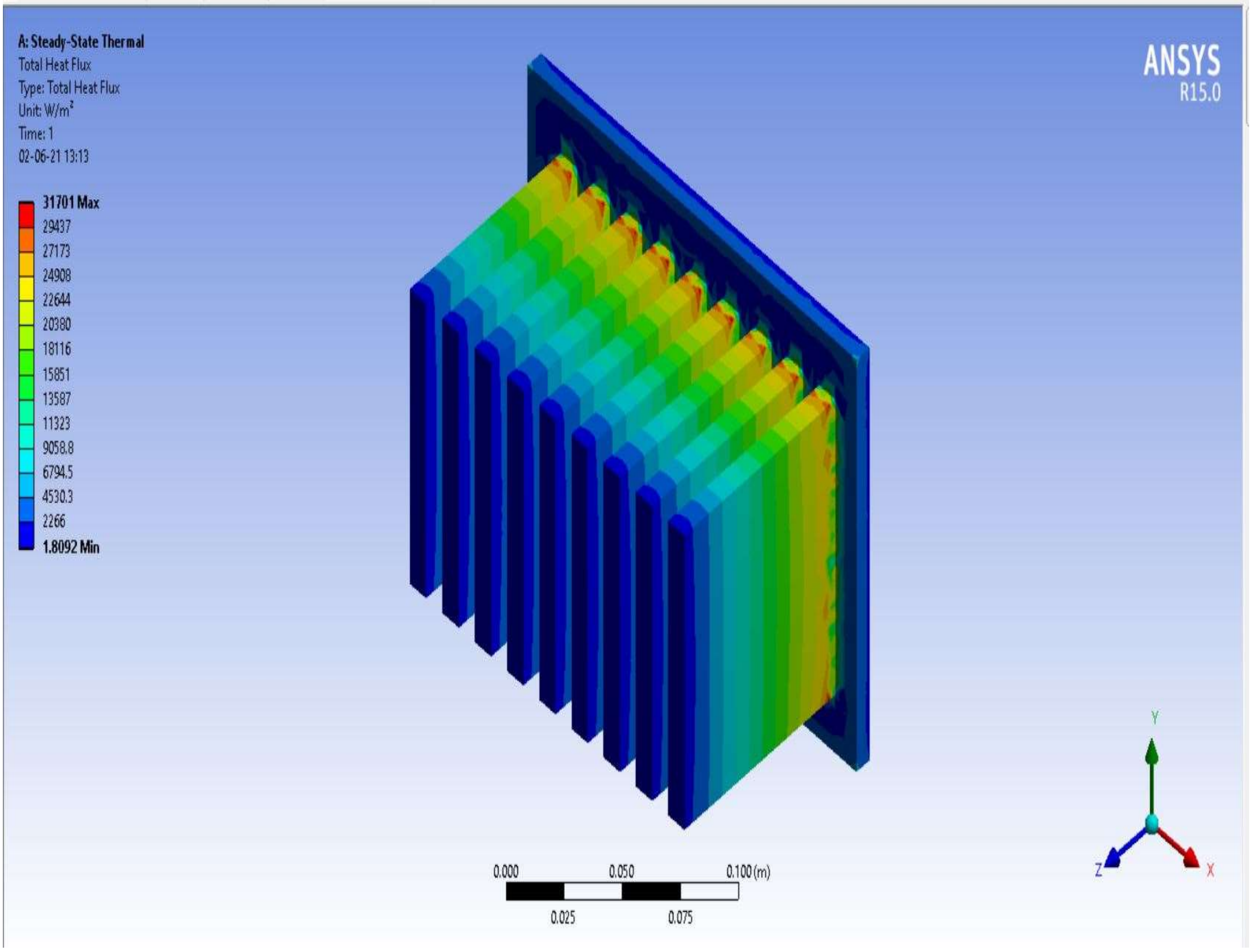


Fig 5.2.2 (a) Heat flux Distribution of Rectangular Fin

(Isometric view)

The total heat flux distribution of orientation to Z-axis is obtained with a maximum of 31701 W/m^2 and minimum of 1.8092 W/m^2. The simulated result is as shown in Fig 5.2.2(a) and Fig 5.2.2(b).

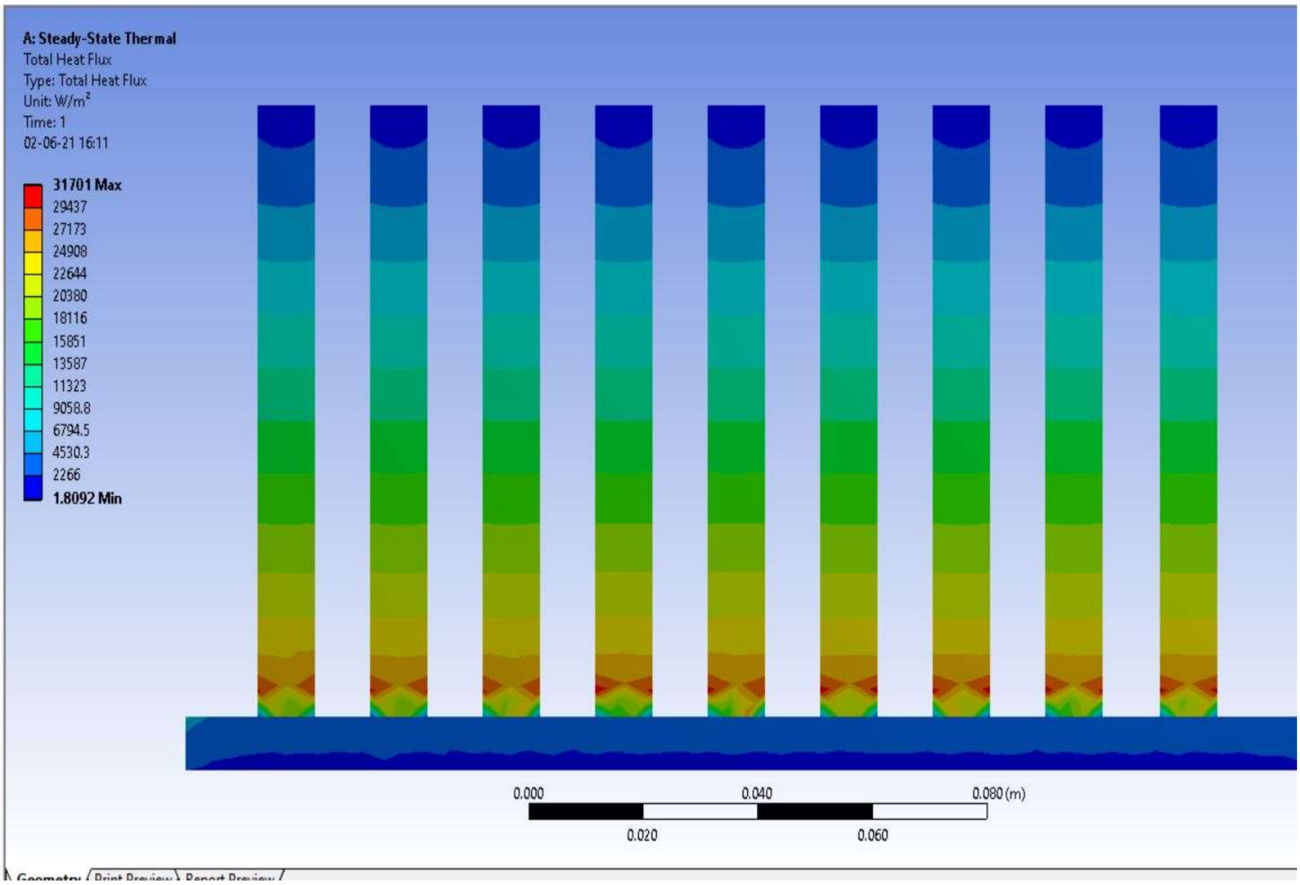


Fig 5.2.2 (b) Heat flux Distribution of Rectangular Fin

(Front view)

5.3 Graphs

Temperature distribution of Steady state thermal analysis is shown in Table 5.

Type of alloy	**Maximum Temperature (º c)**	**Minimum Temperature (º c)**
Aluminum 6063	**200**	**192.53**
Aluminum 7068	**200**	**193.28**

Table 5 Temperature distribution of Steady state thermal analysis

A graph is plotted between the Type of aluminum alloy (on x-axis) and the Temperature distribution of Rectangular fins (on y-axis)

Temperature distribution of Rectangular fins shown in Fig 5.3.1.

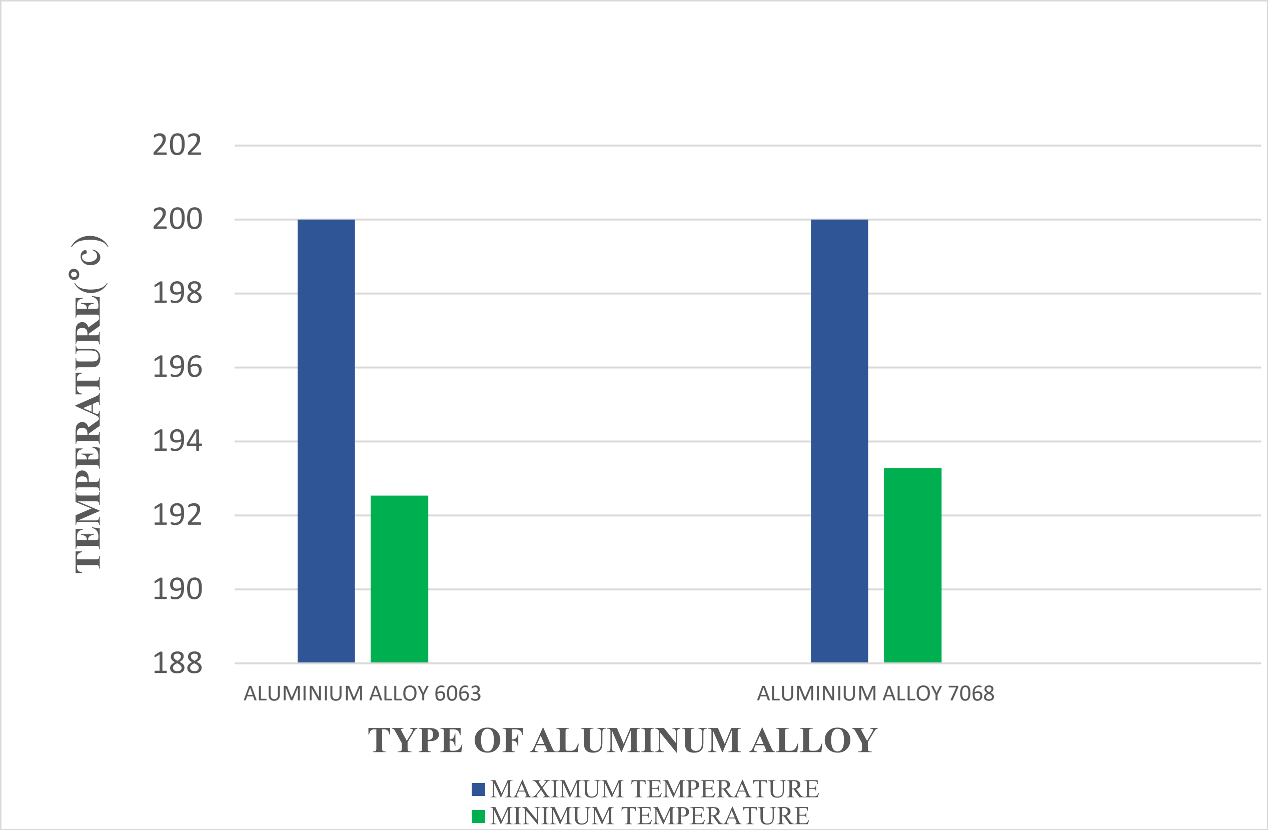


Fig 5.3.1 Temperature distribution of Rectangular fins

Heat flux distribution of Steady state thermal analysis is shown in Table 6.

Type of alloy	Maximum Heat flux	Minimum Heat flux
Aluminum 6063	**36794**	**2.7462**
Aluminum 7068	**31701**	**1.8092**

Table 6 Heat flux distribution of Steady state thermal analysis

A graph is plotted between the Type of aluminum alloy (on x-axis) and the Heat flux distribution of Rectangular fins (on y-axis).

Heat flux distribution of Rectangular fins is shown in Fig 5.3.2.

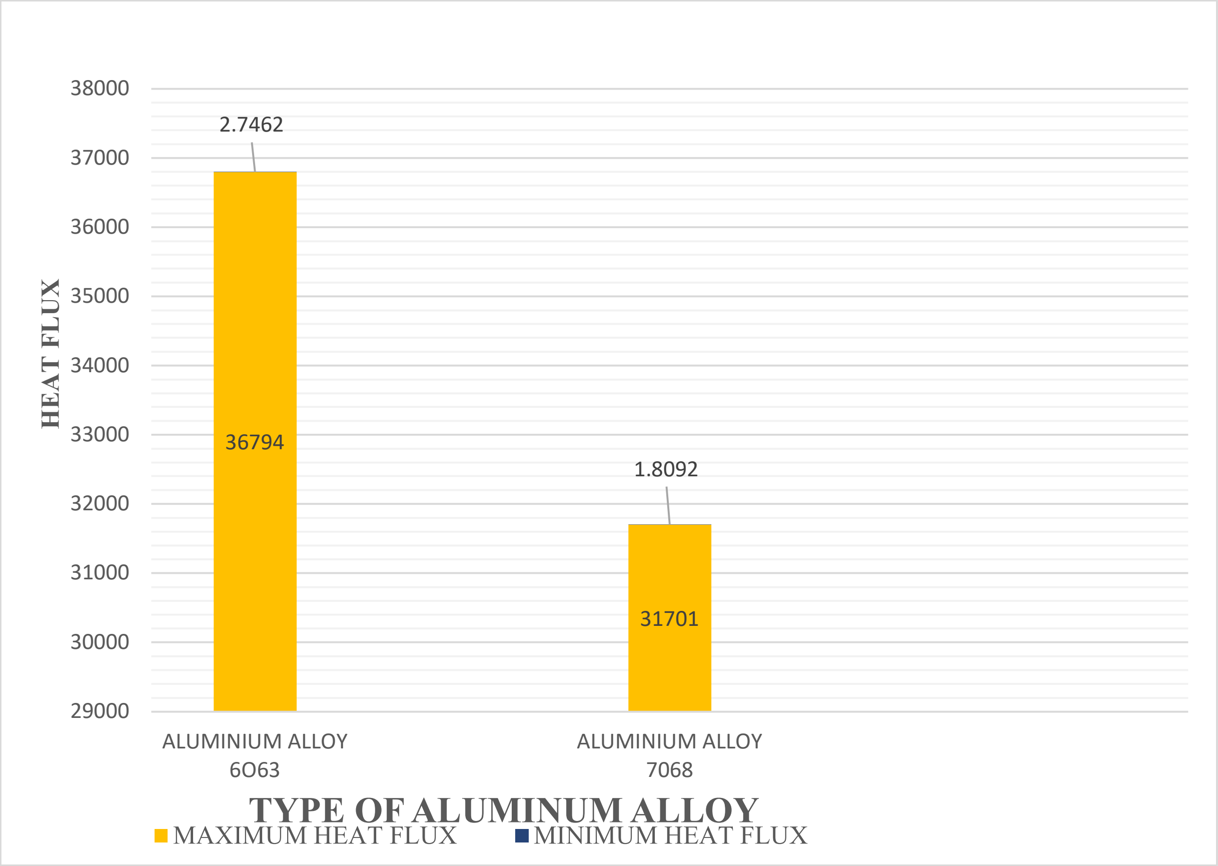


Fig 5.3.2 Heat flux distribution of Rectangular fins

Temperature distribution along specified path of AA 6063 is shown in Fig 5.3.3.

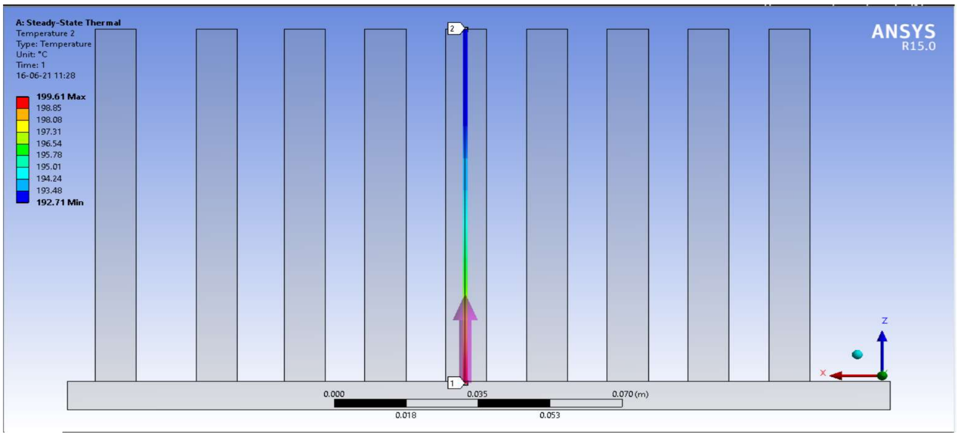


Fig 5.3.3 Temperature distribution along specified path of AA 6063

Temperature distribution along length of AA 6063 is shown in Table 7.

LENGTH	TEMPERATURE
0	199.61
9.1364e-003	198.56
1.8273e-002	197.46
2.7409e-002	196.46
3.6545e-002	195.58
4.5682e-002	194.82
5.4818e-002	194.17
6.3955e-002	193.65
7.3091e-002	193.24
8.2227e-002	192.94
9.1364e-002	192.77
0.1005	192.71

Table 7 Temperature distribution along length of AA 6063

Temperature distribution along length of AA 6063 is shown in Fig 5.3.4.

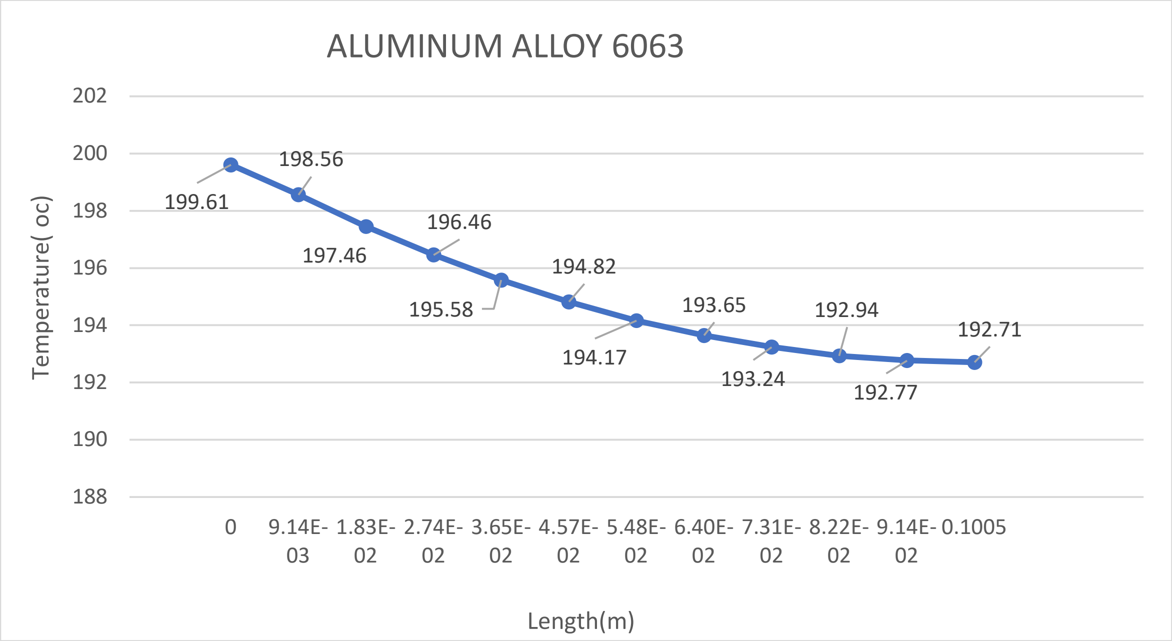


Fig 5.3.4 Temperature distribution along length of AA 6063

Temperature distribution along specified path of AA 7068 is shown in Fig 5.3.5.

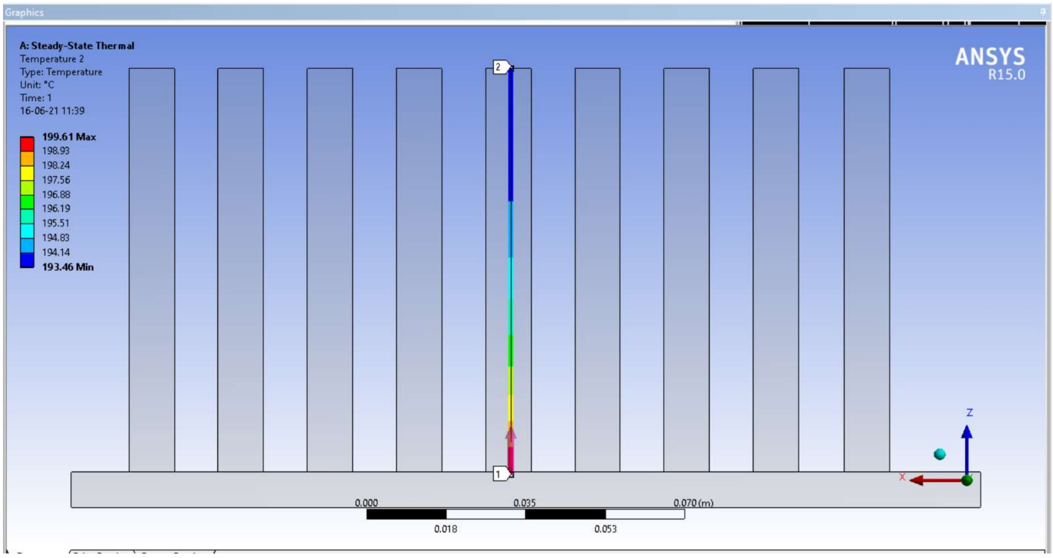


Fig 5.3.5 Temperature distribution along specified path of AA 7068

Temperature distribution along length of AA 7068 is shown in Table 8.

LENGTH	TEMPERATURE
0	199.61
8.4091e-003	198.7
1.6818e-002	197.7
2.5227e-002	196.82
3.3636e-002	196.03
4.2045e-002	195.35
5.0455e-002	194.77
5.8864e-002	194.3
6.7273e-002	193.93
7.5682e-002	193.67
8.4091e-002	193.51
9.25e-002	193.46

Table 8 Temperature distribution along length of AA 7068

Temperature distribution along length of AA 7068 is shown in Fig 5.3.6

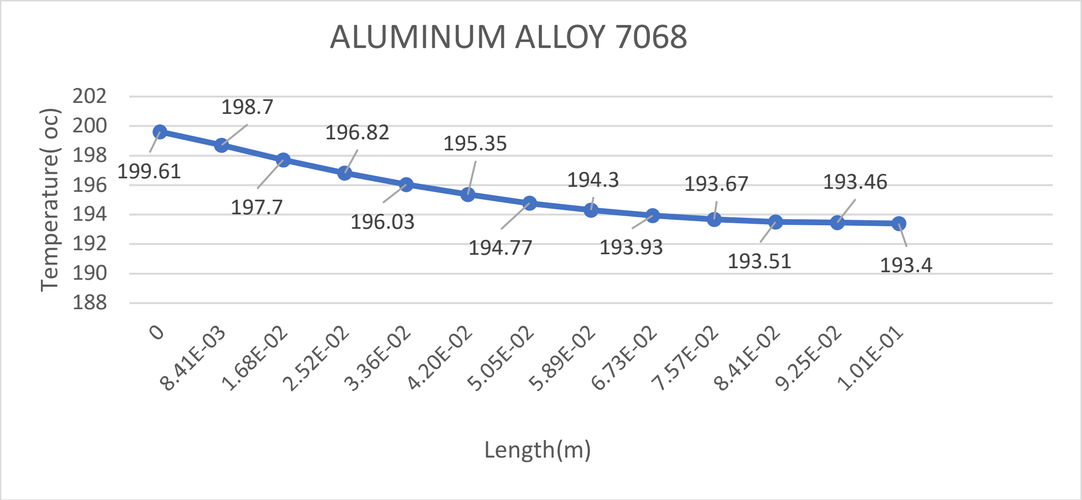


Fig 5.3.6 Temperature distribution along length of AA 7068

From the above graphs it is clear that temperature at the end of Aluminum alloy 6063 fin is minimum, as compare to other fin. If temperature at end of fin is minimum then we can conclude that the aluminum alloy 6063 dissipated maximum amount heat.

From the results obtained by analysis, it has been found that for a particular chosen material of aluminum and a constant convective heat transfer coefficient, the order in which heat dissipation occurs among the considered Rectangular configuration is as below.

ALUMINUM ALLOY 6063 > ALUMINUM ALLOY 7068

CHAPTER 6

CONCLUSION AND FUTURE SCOPE

6.1 Conclusions

In this study, a steady state thermal analysis has been carried out on Fins by varying the material of fins. The results are evaluated on the basis of temperature distribution and heat flux distribution which indicate the total heat dissipation from the entire surface under a fixed volume condition. The thermal performance of Rectangular fin having material of Aluminum alloy 6063 and Aluminum alloy 7068 are compared for the fixed base plate dimensions and fin height under fixed volume conditions.

According to the results obtained, we can conclude that the rectangular fin having material of Aluminum alloy 6063 show better thermal performance in the most practical regions, especially in the process of natural convection. Rectangular fin array usually has larger surface area compared to other considered fins (of different geometries) and it is easy to design. So therefore, the Rectangular fin is considered which dissipates the most heat from the equipment in the case of total heat dissipation (especially in the case of natural convection).

Therefore, it is recommended to use rectangular fin heat having material Aluminum alloy 6063 the total heat dissipation for a given volume of fin is maximized under a fixed volume condition.

6.2 Future scope

In the present study, for the considered material of the fin, chosen material of aluminum and a derived constant convective heat transfer coefficient of 7.614 $W/m^2\ ^0C$, the rectangular fin having material of Aluminum alloy 6063 served the purpose of total heat dissipation, way better compared to fin other material Aluminum alloy 7068. The future work of this study may include modifications of design parameters of fins, orientation of fin, application of through holes etc. The considered modifications with respect to design and orientation can be optimized using various optimization techniques and finally a fin with desirable fin dimensions, material and properties can be obtained to yield better results (high heat dissipation) to enhance the life of expensive electronic components.

REFERENCES

[1] C. Han-Taw and Cs (2006). "Investigation of natural convection heat transfer coefficient on a vertical square fin of finned tube heat exchangers". International Journal of Heat and Mass Transfer 49(1718):3034-3044

[2] C. W. Leung and S. D. Probert "Heat-exchanger performance: influence of gap width between consecutive vertical rectangular fin arrays. Applied Energy. 56: 1-8,1997.

[3] A.Aziz, F.Khani convection-radiation from a continuously moving fin of a variable thermal conductivity.

[4] Arularasan R and Velraj R "CFD analysis in a heat sink for cooling electronic devices", International Journal of the computer, the internet and management, Volume 6, pp.1-11,2008.

[5] H. H. Jung, and J. G. Maveety "Pin fin heat sink modeling and characterization", Sixteenth IEEE Semi-Therm Symposium, IEEE Publications, Piscataway, NJ, pp. 260–265, March2002.

[6] Ahmed F. Khudheyer, Zaid Hameed Hasan "Effect of the fins configuration on natural convection heat transfer experimentally and numerically" International Journal Of Energy And Environment Volume 6, Issue 6, pp.607-628,2015.

[7] Arularasan R and Velraj R "CFD Analsyis In A Heat Sink for Cooling of Electronic Devices" International Journal of The Computer, the Internet and Management Vol. 16. No.3 (September-December 2008) pp 1-11.

[8] G. Hetstoni, A. Mosyak, Z. Segal, G. Ziskind "A uniform temperature heat sink for cooling of electronic devices", International Journal of Heat and Mass transfer, 3275-3286,2002.

[9] Bar-Cohen, A. and Iyengar M "Design and optimization of air-cooled heat sinks for sustainable development", IEEE transactions on components and packaging technologies, Vol. 25(4), pp 584-591, 2002.

[10] Ankit Girolla, Sunil.V.Dingare and Pramod.S.Purandare “A review on the effect of shape on performance of pin fins”, International Journal of Current Engineering and Technology, 12 March 2017.

[11] Shivdas S. Kharche “Heat Transfer Analysis through Fin Array by Using Natural Convection” International Journal of Emerging Technology and Advanced Engineering, Volume 2, 2014.

[12] CP Kothandaraman, S. Subramanyan “Heat and Mass Transfer Data Book”, New Age International Limited(p).

[13] Amit Md. Estiaque Arefin, “Thermal Analysis of Modified Pin Fin Heat Sink for Natural Convection”, 2016 5th International Conference on Informatics, Electronics and Vision (ICIEV).

[14] M. Bahrami, “Natural Convection”, Simon Fraser University ENSC 388 (F09).

[15] Sunil S, Gowreesh S S, Veeresh B R, “Heat Transfer Enhancement and Thermal Performance of Extended Fins”, International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-5, Issue-5, June 2016

[16] Varun R. Yadav, Karan S.Vishe, “Experimental Analysis to Investigate the Thermal Performance of Different Types of Fin Geometry”, International Research Journal of Engineering and Technology (IRJET), Volume: 06 Issue: 02 | Feb 2019

[17] Akshendra Soni, “Study of Thermal Performance between Plate-fin, Pin-fin and Elliptical Fin Heat Sinks in Closed Enclosure under Natural Convection”, International Advanced Research Journal in Science, Engineering and Technology ISO 3297:2007 Certified Vol. 3, Issue 11, November 2016

[18] R. Sam Sukumar , G.Sriharsha, “Modelling And Analysis Of Heat Sink With Rectangular Fins Having Through Holes”, International Journal of Engineering Research and Applications (IJERA), Vol. 3, Issue 2, March - April 2013, pp.1557-1561

[19] V. Himachandra Raju, Ch. Srinivasa Rao “Steady State Thermal Analysis Heat Sink with Rectangular Pin Fin” International Journal for Researchin Applied Science & Engineering Technology (IJRASET), Volume 6, Issue IV, April 2018.