COMBINED NUMERICAL AND FINITE ELEMENTAL THERMAL ANALYSIS OF THE EXTENDED SURFACES USED IN IC ENGINES

A PROJECT REPORT

Submitted by

SHURENDHER KUMAR S (Reg. No.: 211415114300)

SIVAKUMAR P (Reg. No.: 211415114302)

JAI KRISHNA S (Reg. No.: 211415114142)

SIVARAMAKRISHNAN K (Reg. No.: 211415114303)

In partial fulfillment of the award of the degree

of

BACHELOR OF ENGINEERING

in

MECHANICAL ENGINEERING

PANIMALAR ENGINEERING COLLEGE, CHENNAI

ANNA UNIVERSTY: CHENNAI 600 025

APRIL 2019

BONAFIDE CERTIFICATE

Certified that this project report "**COMBINED NUMERICAL AND FINITE ELEMENTAL THERMAL ANALYSIS OF THE EXTENDED SURFACES USED IN IC ENGINES**" is the bonafide work of "**SHURENDHER KUMAR S (Reg. No.:211415114300), SIVAKUMAR P (Reg. No.:211415114302), JAI KRISHNA S (Reg. No.:211415114142), SIVARAMAKRISHNAN K (Reg.No.:211415114303)**" who carried out the project work under my supervision.

SIGNATURE	SIGNATURE
Dr. L.KARTHIKEYAN, M.E, M.B.A, Ph.D.	**Mr. N.BALAJI, M.E.**
PROFESSOR/ HEAD,	**SUPERVISOR,**
Dept. of Mechanical Engineering,	**ASSISTANT PROFESSOR,**
Panimalar Engineering College,	Mechanical department,
Bangalore trunk road,	Panimalar Engineering College,
Poonamallee,	Bangalore trunk road,
Nazarethpettai,	Nazarethpettai,
Chennai- 600 123.	Poonamallee,
	Chennai- 600 123

Submitted for Anna University viva-voce held on _______________ during the year ________________.

INTERNAL EXAMINER EXTERNAL EXAMINER

ACKNOWLEDGEMENT

We wish to express our sincere thanks to our Founder and chairman, **Dr. JEPPIAR**, **M.A., B.L., Ph. D**, for his endeavor in educating me in premier Institution.

We would like to express deep gratitude to our Secretary and correspondent, **Dr. P. CHINNADURAI**, **M.A., M. Phil, B. Ed, Ph. D**, for his kind words and enthusiastic motivation which inspired us a lot in completing this project.

We express my sincere thanks our director **Mrs. C. VIJAYA RAJESHWARI**, and **Mr. C. SHAKTHI KUMAR**, **M.E**., for providing us with necessary facilities for completion of this project.

We would like to express our gratitude to our principal, **Dr. K. MANI**, **M.E., Ph. D.**, for his encouragement and sincere guidance.

We wish to convey my thanks and gratitude to **Dr. L. KARTHIKEYAN**, **M.E., M.B.A., Ph.D.**, Head of the Department of Mechanical Engineering and our dedicated guide for providing us with sample time and encouraging for successful completion of our project.

We are obliged to thanks our internal guide **Mr. N.BALAJI**, **M.E.**, Assistant Professor, Department of Mechanical Engineering for his valuable guidance, support and suggestions.

Finally, we dedicate our efforts to our divine parents who have given us the opportunity to receive education and provide us sample resource and environment to work effectively.

ABSTRACT

The Engine cylinder is one of the major automobile components, which is subjected to high temperature variations and thermal stresses. In order to cool the cylinder, fins are provided on the surface of the cylinder to increase the rate of heat transfer. We know that, by increasing the surface area we can increase the heat dissipation rate, so designing such a large complex engine is very difficult. The main aim of this project is to analyze and increase the heat transfer by varying geometry of fins using **numerical** method and **Finite elemental analysis** using **Ansys work bench**. The calculations were made by applying the boundary conditions to the heat transfer from fin equations. The 3D model of the geometries are created using CATIA V5 and its thermal properties are analyzed using Ansys workbench R 2014. The accurate thermal simulation could permit critical design parameters to be identified for improved life. Presently Material used for manufacturing cylinder fin body is Aluminium Alloy **Al 6082 T6** which has thermal conductivity of 180 W/m°C. The analysis is carried out for various shapes of fins using this material. Based on this analysis, the **optimized design** is made and it is further analyzed in numerical and finite elemental method to determine its performance.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

SL. NO.	SYMBOL	NAME	EXPLANATION
1.	^{o}C	Degree Celsius	It is used to measure the temperature in SI units
2.	W	Watt	It is the measure of energy transfer
3.	K	Kelvin	Temperature is measured $K = 273 + {}^{o}C$
4.	m	Meter	It is used to measure the length or displacement in SI units
5.	µ	Dynamic viscosity	It is the measurement of the fluid's internal resistance to flow
6.	ν	Kinematic viscosity	It is the ratio of dynamic viscosity to density
7.	K	Kilo	It refers to 10^{3}
8.	g	Gram	It is used to measure the weight
9.	mm	Milli meter	It is the measure of distance or displacement in 10^{-3}

CHAPTER-1

INTRODUCTION

1.1. HEAT TRANSFER IN IC ENGINES

We know that in case of Internal Combustion engines, combustion of air and fuel takes place inside the engine cylinder and hot gases are generated. The temperature of gases will be around 2300-2500°C. This is a very high temperature and may result into burning of oil film between the moving parts and may result it seizing or welding of same. So, this temperature must be reduced to about 200-350°C at which the engine will work most efficiently. Too much cooling is also not desirable since it reduces the thermal efficiency. So, the object of cooling system is to keep the engine running at its most efficient operating temperature. It is to be noted that the engine is quite inefficient when it is cold and hence the cooling system is designed in such a way that it prevents cooling when the engine is warming up and till it attains to maximum efficient operating temperature, then it starts cooling. To avoid overheating, and the consequent ill effects, the heat transferred to an engine component (after a certain level) must be removed as quickly as possible and be conveyed to the atmosphere. It will be proper to say the cooling system as a temperature regulation system. It should be remembered that abstraction of heat from the working medium by way of cooling the engine components is a direct thermodynamic loss. This loss is achieved by utilizing coolants. The cooling systems are of two types

1.2. CLASSIFICATION OF COOLING SYSTEM

- Liquid cooled system
- Air cooled system

1.2.1. LIQUID COOLED SYSTEM

In liquid cooled systems, coolants such as water, ethylene glycol, propylene glycol, anti freeze, etc….. This coolant on contact with the heated surface absorbs heat and is then dissipated through a fan-belt system. This type of cooling systems is mostly used in modern cars since the engine is not in direct contact with the atmospheric air and hence air cannot effectively cool the engine. The process involved in liquid coolant system is shown in fig.1.1. This system is complex and involves many components like water pump, thermostat, radiator, fan, pressure cap, hoses and coolant. This can increase the cost of the engine assembly.

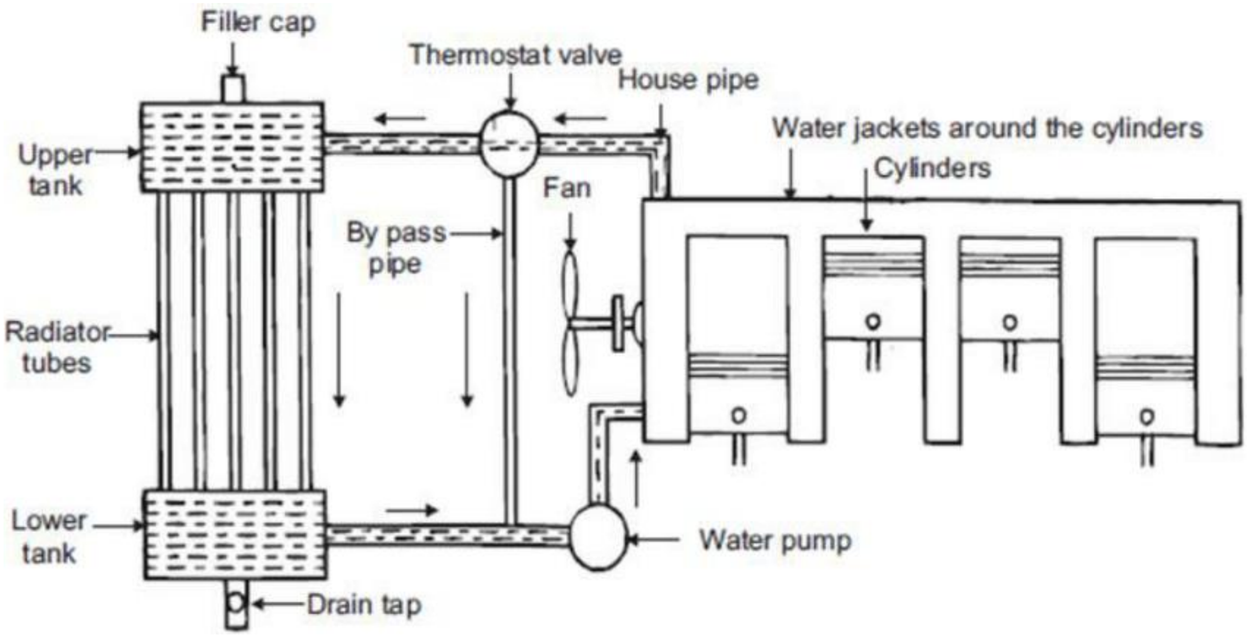


Fig.1.1. Liquid cooled system

1.2.2. AIR COOLED SYSTEM

The other type, air cooled system in comparison is less complex. In this atmospheric air is used as coolant. Thus eliminating the use of special coolants and other flow regulating mechanisms. This type of cooling system is most commonly used in two wheeler engine where abundant amount of air is in contact with the engine. The performance is further increased by adding extended surfaces which serves as additional surfaces for cooling. A typical air cooled engine with an extended surface is shown in fig.1.2. In this project the air cooled system is taken into consideration for optimization. Fins are basically mechanical structures which are used to cool various structures via the process of convection. Most part of their design is basically limited by the design of the system. But still certain parameters and geometry could be modified to better heat transfer. In most of the cases simple fin geometry is preferred such as rectangular fins and circular fins. Many experimental works has been done to improve the heat release of the internal combustion engine cylinder and fin efficiency.

Fig.1.2. Air cooled system

In our analysis air cooled IC Engine with the following specifications is used.

Table 1.1. Cylinder specifications

Bore Diameter	40 mm
Length of Bore	120 mm
Material used	Aluminium alloy 6082 T6
Type of fuel	Petrol

1.3. ALUMINIUM ALLOY 6082

Traditionally, cast iron has been used for manufacture of engine cylinder heads. But recent trends for low weight to power demand have urged the manufacturers to consider using aluminium and magnesium alloy. Aluminium alloy 6082 T6, is considered for analysis in our project.

Table 1.2. Chemical composition of Al 6082

Chemical Element	% Present
Manganese (Mn)	0.40 - 1.00
Iron (Fe)	0.0 - 0.50
Magnesium (Mg)	0.60 - 1.20
Silicon (Si)	0.70 - 1.30
Copper (Cu)	0.0 - 0.10
Zinc (Zn)	0.0 - 0.20
Titanium (Ti)	0.0 - 0.10
Chromium (Cr)	0.0 - 0.25
Other (Each)	0.0 - 0.05
Others (Total)	0.0 - 0.15
Aluminium (Al)	Balance

Among aluminium alloys, Al 6000 series was preferred because of its high corrosion resistance. Aluminium alloy 6082 is a medium strength alloy with excellent corrosion resistance. It has the highest strength of the 6000 series alloys. Alloy 6082 is known as a structural alloy. In plate form, 6082 is the alloy most commonly used for machining. As a relatively new alloy, the higher strength of 6082 has seen it replace 6061 in many applications. The addition of a large amount of manganese controls the grain structure which in turn results in a stronger alloy.

The extruded surface finish is not as smooth as other similar strength alloys in the 6000 series. In the T6 and T651 temper, alloy 6082 machines well and produce tight coils of swarf when chip breakers are used.

Table 1.3. Thermal properties of Al 6082

Physical Property	Value
Density	2700 Kg/m^3
Melting Point	555 °C
Thermal Expansion	24 $x10^{-6}$ /°C
Thermal Conductivity	180 W/m°C

CHAPTER-2

HEAT TRANSFER

Heat transfer is the process of transfer of heat from high temperature reservoir to low temperature reservoir. In terms of the thermodynamic system, heat transfer is the movement of heat across the boundary of the system due to temperature difference between the system and the surroundings. The heat transfer can also take place within the system due to temperature difference at various points inside the system. The difference in temperature is considered to be ‘potential’ that causes the flow of heat and the heat itself is called as flux. Heat is a form of energy which transfers between bodies which are kept under thermal interactions.

2.1. MODES OF HEAT TRANSFER

Transfer of heat occurs in three modes. Three modes of heat transfer are described below. When a temperature difference occurs between two bodies or a body with its surroundings, heat transfer occurs.

- Conduction
- Convection and
- Radiation

2.2. CONDUCTION HEAT TRANSFER

Conduction takes place at a microscopic level. Atoms or molecules at higher temperature have high levels of energy. Through vibration, this energy is passed on to neighbouring atoms and molecules. In other words, in conductive mode of heat transfer, vibrating atoms and molecules a part of their energy. This kind of heat transfer can take place between two or more substances or through the substance. Conduction can also take place when electrons move from one atom to another. Transient conduction takes place when temperature within an object changes the function of time while steady state conduction involves no change in temperature with time. In short, conduction is the process of transfer of heat between the same medium. The rate of heat transfer through conduction is governed by the Fourier's law of heat conduction.

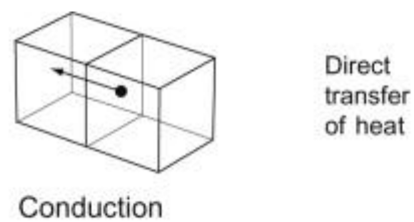


Fig.2.1. Conduction heat transfer

2.2.1. FOURIER'S LAW OF HEAT CONDUCTION

As mentioned above, the rate of heat transfer through conduction is governed by Fourier's law of heat conduction. According to Fourier's law, the rate of heat flow, q , through a homogeneous solid is directly proportional to the area A, of the section at the right angles to the direction of the heat flow, and to the temperature difference ∇T along the path of heat flow. Mathematically, it can be written as

$Q = -kA(dT/dx)$

Where, 'Q' is the heat flow rate by conduction,

'K' is the thermal conductivity of body material

'A' is the cross-sectional area normal to direction of heat flow

'dT/dx' is the temperature gradient of the section.

The figure shown below illustrates the Fourier law of heat conduction.

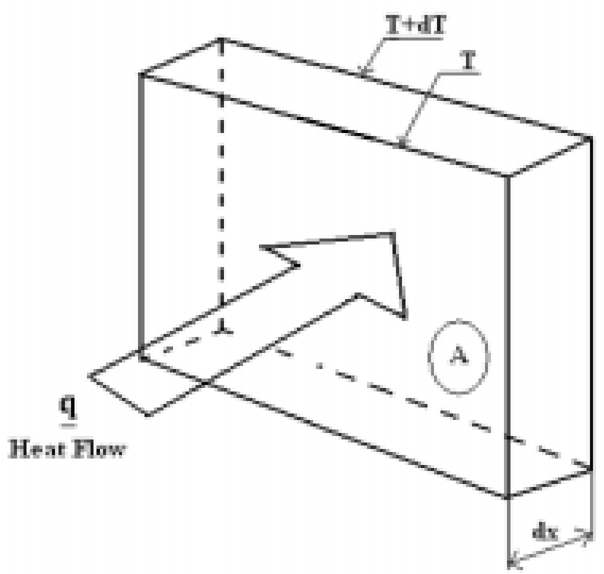


Fig.2.2. Heat flow through a homogeneous (isotropic) solid

2.3. CONVECTIVE HEAT TRANSFER

Convection is a mode of heat transfer which takes place through the movement of collective masses of heated atoms and molecules. Convection requires actual flow of material particles whereas in conduction, the heat is transferred through vibration without the atoms or molecules leaving their original position. In convection, heat transfer takes place through both diffusion and advection. As convection requires the actual movement of the heated atoms/ molecules, it requires presence of a fluid for heat transfer. In short, convective heat transfer involves transfer of heat between two different mediums as shown in figure 2.3. The rate of convective heat transfer is governed by the Newton's law of cooling.

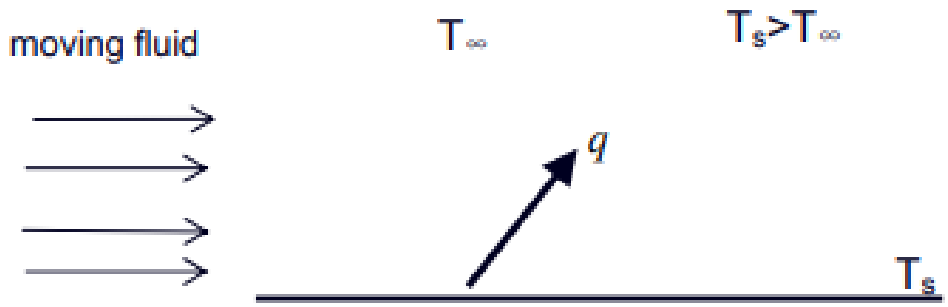


Fig.2.3. Convective heat transfer

2.3.1. NEWTON'S LAW OF COOLING

Newton's law of cooling states that, "The rate of cooling is directly proportional to the mean difference in temperature between the body and the surroundings.

$q = h A_s \Delta T$

$q" = h(T_s - T_\infty)$

Where, q = heat flow from surface

h = heat transfer coefficient

A_s = Surface area from which convection is occurring.

T_s = Surface Temperature

T_∞ = Ambient Temperature

$\Delta T = T_S - T_\infty$ = Temperature Difference between surface and coolant.

$q"$ = Heat flux = q/A_s

It is to be noted that heat transfer co efficient, h is not a thermodynamic property of the material, but may depend on geometry of surface, flow characteristics, thermodynamic properties of the fluid, etc. Typical range of values of h is shown in table 2.1.

Table 2.1. Typical range of ‘h’ (W/m^2 ^{o}C)

Heat transfer type	Value	
	Gases	Liquids
Free convection	2 - 25	50 – 100
Forced convection	25 - 250	50 - 20,000
Boiling/Condensation	2500 -100,000	

2.3.2. TYPES OF CONVECTION

Convection → Free or natural convection (Induced by buoyancy forces)

Convection → Forced convection (Induced by external means)

May occur with phase change (boiling, condensation)

In this project free convection is assumed for analysis.

2.3.3. DIMENSIONLESS NUMBERS

REYNOLDS NUMBER (Re)

The Reynolds number (Re) is an important dimensionless quantity in fluid mechanics used to help predict flow patterns in different fluid flow situations. At low Reynolds numbers, flows tend to be dominated by laminar (sheet-like) flow, while at high Reynolds numbers turbulence results from differences in the fluid's speed and direction, which may sometimes intersect or even move counter to the overall direction of the flow (eddy currents). These eddy currents begin to churn the flow, using up energy in the process, which for liquids increases the chances of cavitations. In short, Reynolds's number is defined as the ratio inertia force (ρ u L) to viscous or friction force (μ).

Reynolds Number can therefore be expressed as

$Re = \rho\ u\ L / \mu$

$= \rho\ u^2 / (\mu\ u / L)$

$= u\ L / \nu$

Where

Re = Reynolds Number (non-dimensional)

ρ = density (kg/m^3, lb_m/ft^3)

u = velocity based on the actual cross section area of the duct or pipe (m/s, ft/s)

μ = dynamic viscosity (Ns/m^2, $lb_m/s\ ft$)

L = characteristic length (m, ft)

$\nu = \mu / \rho$ = kinematic viscosity (m^2/s, ft^2/s)

The Reynolds Number can be used to determine if flow is laminar, transient or turbulent. The flow is

- laminar - when Re < 2300
- transient - when 2300 < Re < 4000
- turbulent - when Re > 4000

In practice laminar flow is only actual for viscous fluids - like crude oil, fuel oil and other oils.

PRANDTL NUMBER (Pr)

Prandtl number, Pr, is a dimensionless parameter representing the ratio of diffusion of momentum to diffusion of heat in a fluid. Prandtl number is a characteristic of the fluid only. For air at room temperature Pr is 0.71 and most common gases have similar values. The Prandtl number of water at 17°C is 7.56. Liquids in general have high Prandtl numbers, with values as high as 10^5 for some oils.

$$Pr = \frac{\nu}{\kappa} = \frac{\eta/\rho}{\lambda/\rho c_p} = \frac{\eta c_p}{\lambda}$$

Where

ν is kinematic viscosity

κ thermal diffusivity.

NUSSELT NUMBER (Nu)

The Nusselt number (Nu) is the ratio of convective to conductive heat transfer across (normal to) the boundary. In this context, convection includes both advection and diffusion. Named after Wilhelm Nusselt, it is a dimensionless number. The conductive component is measured under the same conditions as the heat convection but with a (hypothetically) stagnant (or motionless) fluid. A similar non-dimensional parameter is Biot number, with the difference that the thermal conductivity is of the solid body and not the fluid.

A Nusselt number close to one, namely convection and conduction of similar magnitude, is characteristic of "slug flow" or laminar flow. A larger Nusselt number corresponds to more active convection, with turbulent flow typically in the 100–1000 range. The convection and conduction heat flows are parallel to each other and to the surface normal of the boundary surface, and are all perpendicular to the mean fluid flow in the simple case.

Nu = hL/K (or) hd/K

Where

Nu = Nusselt number

L = characteristic length

d = characteristic diameter

h = heat transfer coefficient

K = Thermal conductivity

2.4. RADIANT HEAT TRANSFER

Radiation is a mode of heat transfer which takes place through vacuum and hence, does not need a physical medium. Radiation takes place either through vacuum or through a transparent medium. In radiative mode, heat transfer takes place through photons present in the electromagnetic waves. The random movement of atoms and molecules in heated substances results in emission of electromagnetic waves which carry the heat to be transferred. The radiative heat transfer is governed by Steffen-Boltzmanlaw.

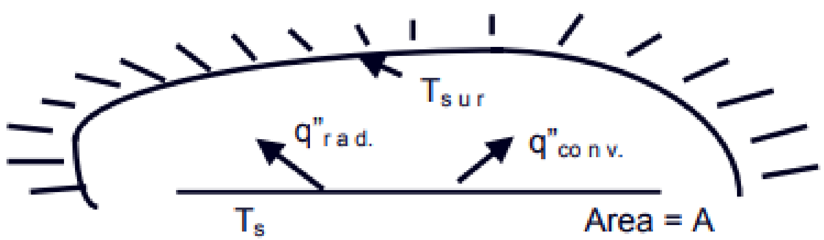


Fig.2.4. Radiant heat transfer

2.4.1. STEFFAN-BOLTZMAN LAW

Steffen-Boltzman law states that, “A body radiates heat at all temperatures above the absolute zero, irrespective of the ambient temperature.” This can be illustrated using fig.2.4.

Emissive power of a surface: $E=\sigma\varepsilon T_s^4$ (W/ m^2)

Where

ε = emissivity which is a surface property (ε = 1 is black body)

σ = Steffen Boltzman constant = 5.67 x 10-8 W/m^2 K^4 .

T_s = Absolute temperature of the surface (K)

The above equation is derived from Steffan Boltzman law, which describes a gross heat emission rather than heat transfer. The expression for the actual radiation heat transfer rate between surfaces having arbitrary orientations can be quite complex. However, the rate of radiation heat exchange between a small surface and a large surrounding is given by the following expression:

$q = \varepsilon \cdot \sigma \cdot A \cdot (T_s^{\,4} - T_{sur}^4)$

Where

ε = Surface Emissivity

A= Surface Area

T_s = Absolute temperature of surface. (K)

T_{sur} = Absolute temperature of surroundings. (K)

In this project, only the effect of conduction and convection is considered for analysis and the effect of radiation is neglected.

CHAPTER-3

EXTENDED SURFACE HEAT TRANSFER

3.1. INTRODUCTION

Convection: Heat transfer between a solid surface and a moving fluid is governed by the Newton's cooling law:

$q = hA_s (T_s - T_\infty)$

Where

T_s is the surface temperature

T_∞ is the fluid temperature

Therefore, to increase the convective heat transfer, one can

- Increase the temperature difference ($T_s - T_\infty$) between the surface and the fluid.
- Increase the convection coefficient h. This can be accomplished by increasing the fluid flow over the surface since h is a function of the flow velocity and the higher the velocity, the higher the h. Example: a cooling fan.
- Increase the contact surface area A. Example: a heat sink with fins.

Many times, when the first option is not in our control and the second option (i.e. increasing h) is already stretched to its limit, we are left with the only alternative of increasing the effective surface area by using fins or extended surfaces. Fins are protrusions from the base surface into the cooling fluid, so that the extra surface of the protrusions is also in contact with the fluid

Example of surfaces where fins are used

1. Air cooled I.C. engines
2. Refrigeration condenser tubes
3. Electric transformers
4. Reciprocating air compressors
5. Semiconductor devices

6. Automobile radiator

In this project, we have discussed about the fins used in automobile air cooled IC engine applications.

3.2. TYPES OF FINS

Fins can be broadly classified as:

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

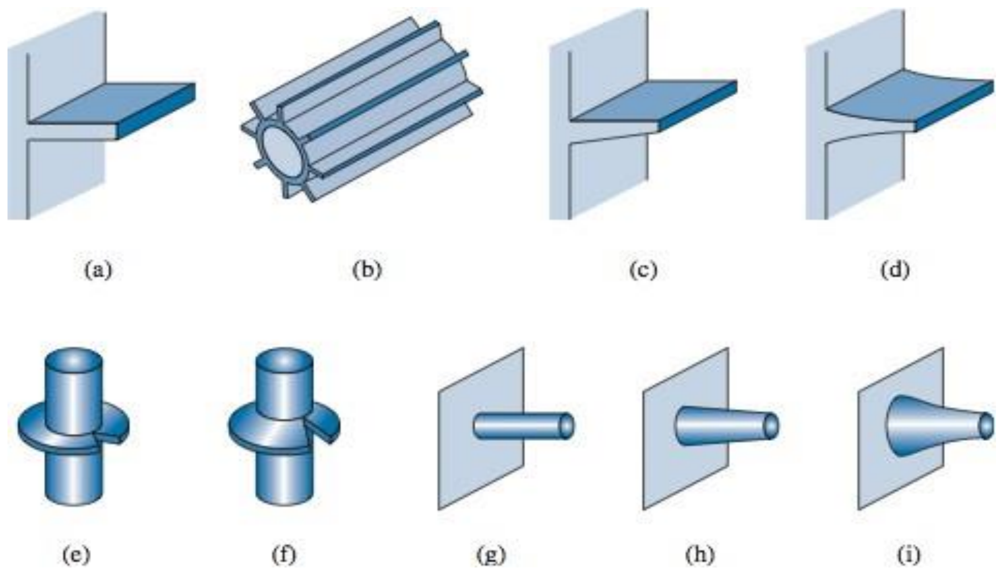


Fig.3.1. Types of fins

3.3. ANALYSIS OF FINS WITH UNIFORM CROSS SECTIONAL AREA

Consider the fig.3.2.The fin is situated on the surface of a hot surface at T_o and surrounded by a coolant at temperature T_∞, which cools with convective coefficient, h. The fin has a cross sectional area, A_c, (This is the area through with heat is conducted.) and an overall length, L. Note that as energy is conducted down the length of the fin, some portion is lost, by convection, from the sides. Thus the heat flow varies along the length of the fin. We further note that the arrows indicating the direction of heat flow point in both the x and y directions. This is an indication that this is truly a two- or three-dimensional heat flow, depending on the geometry of the fin. However, quite often, it is convenient to analyze a fin by examining an equivalent one–dimensional system. The equivalent system will involve the introduction of heat sinks (negative heat sources), which remove an amount of energy equivalent to what would be lost through the sides by convection.

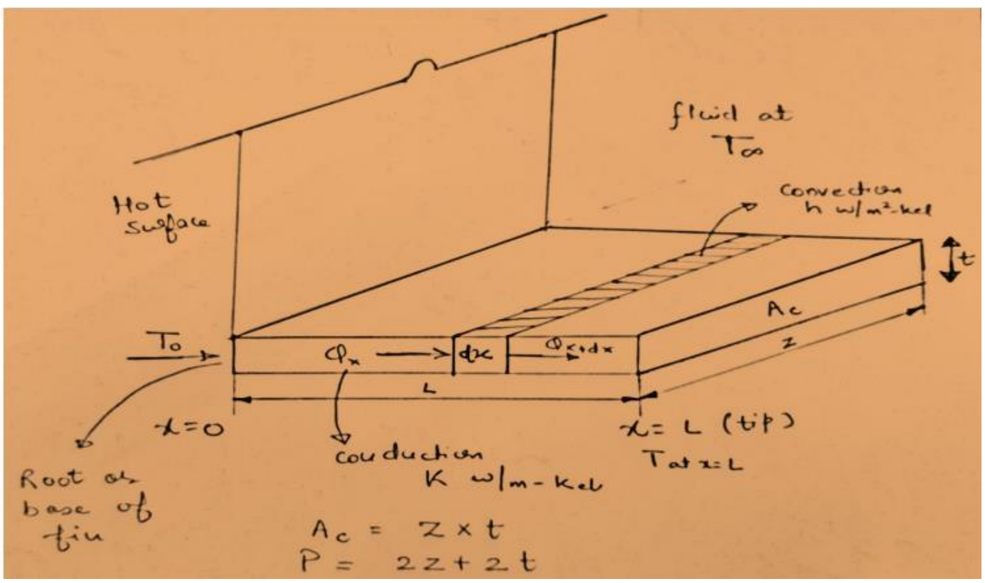


Fig.3.2. Constant area cross section fin

Heat is conducted from the base in to the fin at its root and then while simultaneously conducting along the length of the fin, heat is also convected from the surface of the fin to the ambient fluid with the convective heat transfer coefficient of h in W/m2–Kelvin. Consider a differential element of the fin of length dx.

Let Q_x is the heat conducted in to the element along x- direction given by

$Q_x = -kAc\ dT\ dx$ (from Fourier law of heat conduction) (1)

Where

k = thermal conductivity of fin material

A_c= Area of cross section of the fin

Let Q_{x+dx} = heat conducted out of the element along x-direction

$Q_{x+dx} = Q_x + (Q_x)\, dx$ (2)

$Q_{convected}$ = heat transfer by convection from the surface of element to fluid

$Q_{convected}= h\, (A_{conv})\, (T_s - T_\infty)$ (3)

A_{conv} (convection area) = perimeter of fin length of element

= P x dx

T = temperature of differential element

Assume steady state conditions and writing the energy balance equations for the element

Heat conducted in to the element = heat conducted out of the element + heat convected from the element to fluid

$Q_x = Q_{x+dx} + Q_{convected}$ $Q_x= Q_x + (Q_x)\, dx + h(A_{conv})\, (T_s - T_\infty)$ [from equation1,2and 3]

$0 = б/бx\, (-kA\, dT/dx)dx + h\, (P\, dx)\, (T_s - T_\infty)$

Assuming k constant, we get

$d^2t/dx^2 - hP/KA_c\, (T_s - T_\infty) = 0$

Put $T_s - T_\infty = \Theta$, then

$dT/dx = d\Theta/dx$ and $d^2T/dx^2= d^2\Theta/dx^2$

Put $hP/KA_c = m^2$,

Then $d^2\Theta/dx^2 - m^2\Theta = 0$

This is a standard format of 2nd order differential equation in whose general solution can be given as

$\Theta = C_1e^{-mx} + C_2e^{mx}$

Where $m = \sqrt{(hP/KA_c)}$

And C1 and C2 are constant of integration that are to be obtained from boundary conditions.

Boundary conditions:

One common boundary condition is

At x = 0 (root), $T = T_o$ and $\Theta = \Theta_o = T_o - T_\infty$

The other boundary condition i.e. at the tip depends upon three different cases which are as follows:

Case -1: Fin is infinitely long (very long fin)

When the fin is infinitely long then the temperature at the tip of the fin will be essentially that of the fluid

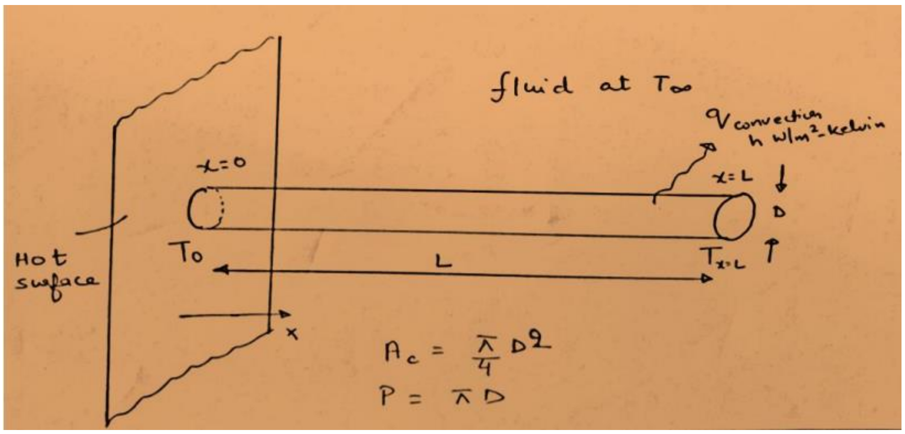


Fig.3.3. Infinite long fin

At $x = \infty$, $\Theta = T - T_\infty = 0$

The general solution is of the form

$\Theta = C_1 e^{-mx} + C_2 e^{mx}$

when $C1 \longrightarrow 0$

$\Theta = C_2 e^{mx}$

On applying boundary condition at root (x=0), we get temperature distribution along fin as

$T = (T_o - T_\infty)\, e^{-mx} + T_\infty$

The heat transfer through fin is:

$Q_{fin} = -kAc\ (dT/dx)_{x=0} = \surd\ (To - T_\infty)$

Case -2: Fin tip is insulated Pin fin

When the fin tip is insulated then

Conduction heat transfer at x = L is equal to zero i.e. $(-kA\, dT/dx)_{x = L} = 0$

Hence, boundary condition will be $(dT/dx)_{x=L} = 0$ and $(d\Theta/dx)x=L = 0$

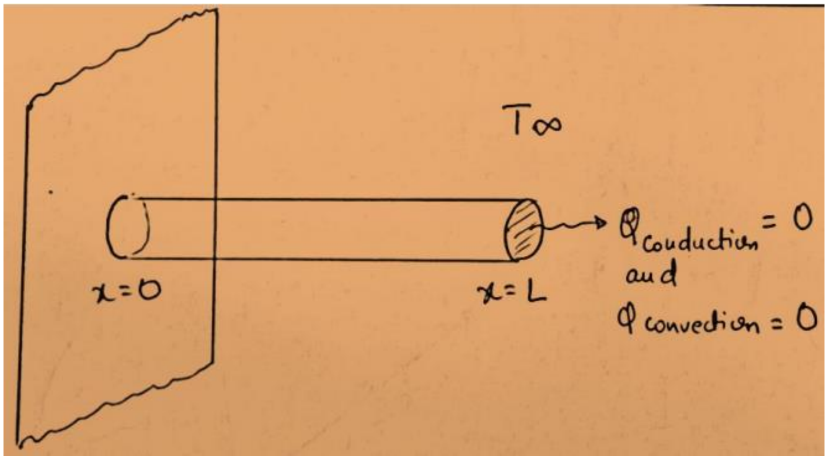


Fig.3.4. Short fin with tip insulated

The temperature distribution along the length of the fin is given by:

$\Theta/ \Theta_o = ((T - T_\infty)/(T_o - T_\infty)) = \text{Cosh}\,[m(L-X)]/\text{Cosh}[mL]$

The resulting heat transfer rate through the fin will be

$Q_{fin}= -kA_c\,(dT/dx)_{x=0} = \sqrt{(hPKA_c)}\,\Theta_o\, \text{Tanh}\, mL$

Case -3: Fin is finite in length and also loses heat by convection from its tip (End not insulated)

Conduction heat transfer at x = L is equal to convection heat transfer from tip

i.e. $(-kA\, dT/dx)_{x = L} = h(Aconv)\,(T_{x=L} - T_\infty)$

Then the temperature distribution is given by

$$\Theta/\Theta_o=((T-T_\infty)/(T_o-T_\infty))=\frac{\text{Cosh}[m(L-X)]+((h/mK)\text{Sinh}\, m(L-X))}{\text{Cosh}(mL)+(h/mK)\text{Sinh}(mL)}$$

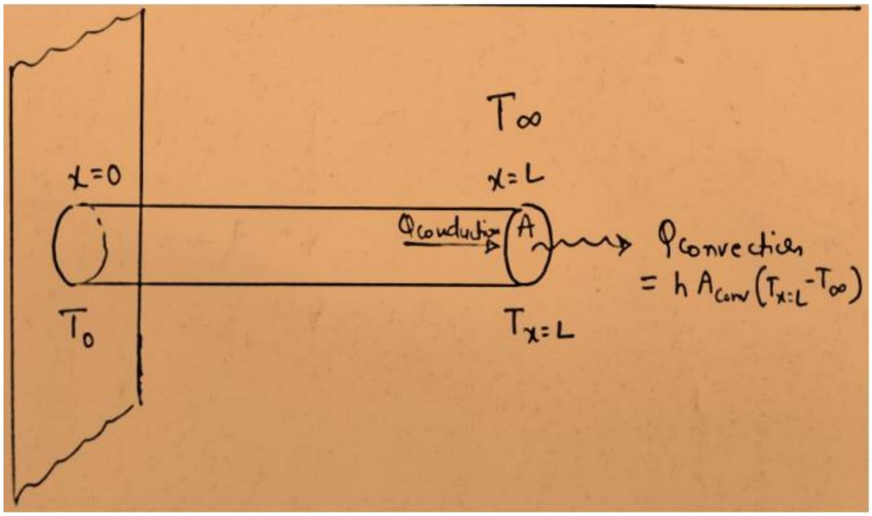


Fig.3.5. Short fin end not insulated

Heat transfer rate through fin is

$$Q_{fin} = (T_b - T_\infty)\ \frac{\tanh(mL) + (h/mK)\ (hPKA)^{0.5}}{1 + (h/mK)\tanh(mL)}$$

Efficiency of fin (η_{fin}):

The efficiency of a fin is defined as the ratio of the actual heat transfer from the fin to that the heat that would be dissipated if whole surface of the fin is maintained at base temperature.

η_{fin} = Actual heat transferred by the fin

According to the definition, efficiency of the fin of infinite length is given as below

$\eta_{fin} = h.P.K.A_c\ (T_s - T_a)/h.A_s\ (T_s - T_a)$

$\eta fin = h.P.K.A_c/h.A_s$

Effectiveness of fin (ε_{fin}):

It is defined as the ratio of the actual heat transfer that takes place from the fin to the heat that would be dissipated from the same surface area without fin.

By above definition ε for infinite length fin is given by

$\varepsilon_{fin} = h.P.K.A_c(T_s\text{-}T_a)/h.A_c(T_s\text{-}T_a)$

Factors affecting fin effectiveness

- $P.Kh.A_c$ should be greater than unity if the rate of heat transfer from the primary surface is to be improved.
- If the ratio of P and Ac is increased, the effectiveness of fin is improved.
- Use of fin will be more effective with materials of large thermal conductivities.

CHAPTER-4

LITERATURE REVIEW

G. Babu and M. Lavakumar analyzed the thermal properties by varying geometry, material and thickness of cylinder fins. The models were created by varying the geometry, rectangular, circular and curved shaped fins and also by varying thickness of the fins. Material used for manufacturing cylinder fin body was Aluminium Alloy 204 which has thermal conductivity of 110-180W/m^{o}C and also using Aluminium alloy 6061 and Magnesium alloy which have higher thermal conductivities. They concluded that by reducing the thickness and also by changing the shape of the fin to curve shaped, the weight of the fin body reduces thereby increasing the efficiency. The weight of the fin body is reduced when Magnesium alloy is used and using circular fin, material Aluminium alloy 6061 and thickness of 2.5mm is better since heat transfer rate is more and using circular fins the heat lost is more, efficiency and effectiveness is also more.

J. Ajay Paul et.al. carried out Numerical Simulations to determine heat transfer characteristics of different fin parameters namely, number of fins, fin thickness at varying air velocities. A cylinder with a single fin mounted Thermal Analysis of Engine Cylinder Fin by Varying Its Geometry and on it was tested experimentally. The numerical simulation of the same setup was done using CFD. Cylinders with fins of 4 mm and 6 mm thickness were simulated for 1, 3, 4 & 6 fin configurations. They concluded that 1. When fin thickness was increased, the reduced gap between the fins resulted in swirls being created which helped in increasing the heat transfer. 2. Large number of fins with less thickness can be preferred in high speed vehicles than thick fins with less numbers as it helps inducing greater turbulence and hence higher heat transfer.

N. Phani Raja Rao et.al. analyzed the thermal properties by varying geometry, material and thickness of cylinder fins. Different material used for cylinder fin were Aluminium Alloy A204, Aluminium alloy 6061 and Magnesium alloy which have higher thermal conductivities and shown that by reducing the thickness and also by changing the shape of the fin to circular shaped, the weight of the fin body reduces thereby increasing the heat transfer rate and efficiency of the fin. The results shows, by using circular fin with material Aluminium Alloy 6061 is better since heat transfer rate, Efficiency and Effectiveness of the fin is more.

Rameshkumar.A and Nandha.S Kumar analyzed the thermal properties of Engine by adopting extensions like rectangular, triangular, trapezium and circular. The analysis was carried out in ANSYS and the results show that fins having extensions have higher rate of heat transfer and an increase of 5-13%.

Abdullah, H. Alessa: studied the natural convection heat transfer enhancement from a horizontal rectangular fin embedded with equilateral triangular perforations. He concluded that, For certain values of triangular dimensions, the perforated fin can result in heat transfer enhancement. The magnitude of enhancement is proportional to the fin thickness and its thermal conductivity. The perforation of fins enhances heat dissipation rates and at the same time decreases the expenditure of the fin material.

Ramdas Pradip: He studied the many industries are utilizing thermal systems whereinn overheating can damage the system components and lead to failure of the system. In order to overcome this problem, thermal systems with effective emitters such as ribs, fins, baffles etc. are desirable. The need to increase the thermal performance of the systems, thereby affecting energy, material and cost savings has led to development and use of many techniques termed as “Heat transfer Augmentation”. This technique is also termed as “Heat transfer

Enhancement" or "Intensification". Augmentation techniques increase convective heat transfer by reducing the thermal resistance in a heat exchanger. Many heat augmentation techniques has been reviewed, these are (a) surface roughness, (b) plate baffle and wave baffle, (c) perforated baffle, (d) inclined baffle, (e) porous baffle, (f) corrugated channel, (g) twisted tape inserts, (h) discontinuous Crossed Ribs and Grooves. Most of these enhancement techniques are based on the baffle arrangement. Use of Heat transfer enhancement techniques lead to increase in heat transfer coefficient but at the cost of increase in pressure drop.

CHAPTER-5

SOFTWRE USED

5.1. CATIA V5

CATIA, an acronym of Computer Aided Three-dimensional Interactive Application) is a multi-platform software suite for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), PLM and 3D, developed by the French company Dassault Systems. Its Initial release was 42 years ago i.e.) 1977. Its types include CAD, CAM, CAE, PLM and 3D

Fig.5.1. Dassault Systems CATIA

5.1.1. SCOPE OF APPLICATION

Commonly referred to as a 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development, including conceptualization, design (CAD), engineering (CAE) and manufacturing (CAM). CATIA facilitates collaborative engineering across disciplines around its 3DEXPERIENCE platform, including surfacing & shape design, electrical, fluid and electronic systems design, mechanical engineering and systems engineering.

CATIA facilitates the design of electronic, electrical, and distributed systems such as fluid and HVAC systems, all the way to the production of documentation for manufacturing.

Mechanical engineering

CATIA enables the creation of 3D parts, from 2D sketches, sheet metal, composites, molded, forged or tooling parts up to the definition of mechanical assemblies. The software provides advanced technologies for mechanical surfacing & BIW. It provides tools to complete product definition, including functional tolerances as well as kinematics definition. CATIA provides a wide range of applications for tooling design, for both generic tooling and mold & die. In the case of Aerospace engineering an additional module named the aerospace sheet metal design offers the user combine the capabilities of generative Sheet metal design and generative surface design.

Design

CATIA offers a solution to shape design, styling, surfacing workflow and visualization to create, modify, and validate complex innovative shapes from industrial design to Class-A surfacing with the ICEM surfacing technologies. CATIA supports multiple stages of product design whether started from scratch or from 2D sketches (blueprints).

Automotive

Many automotive companies use CATIA to varying degrees, including BMW, Porsche, McLaren Automotive, Chrysler, Honda, Audi, Jaguar Land Rover, Volkswagen, SEAT, Skoda, Bentley Motors Limited, Volvo, Fiat, Bentlee International, PSA Peugeot Citroën, Renault, Toyota, Ford, Scania, Hyundai, Tesla Motors, Rolls-Royce Motors, Valmet Automotive, Proton, Elba, Tata motors and Mahindra & Mahindra Limited. Goodyear uses it in making tires for automotive and aerospace and also uses a customized CATIA for its design and development. Many automotive companies use CATIA for car structures – door beams, IP supports, bumper beams, roof rails, side rails, body components because of CATIA's capabilities in Computer representation of surfaces. Bombardier Transportation of Canada is using this software to design its entire fleet of Train engines and coaches. Webasto uses CATIA to design its roof.

Fluid systems

CATIA v5 offers a solution to facilitate the design and manufacturing of routed systems including tubing, piping, Heating, Ventilating & Air Conditioning (HVAC). Capabilities include requirements capture, 2D diagrams for defining hydraulic, pneumatic and HVAC systems, as well as Piping and Instrumentation Diagram (P&ID). Powerful capabilities are provided that enables these 2D diagrams to be used to drive the interactive 3D routing and placing of system components, in the context of the digital mockup of the complete product or process plant, through to the delivery of manufacturing information including reports and piping isometric drawings.

In this project, part modeling system is predominantly used for designing the models and some drafting sub system is used for drafting the model. The process involved in making the models for analysis is explained separately in the upcoming chapters.

5.2. ANSYS V14

Ansys was founded in 1970 by John Swanson. Swanson sold his interest in the 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.

Fig.5.2. ANSYS

5.2.1. 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, fatigue, fractures, fluid flow, temperature distribution, electromagnetic efficiency and other effects over time.

Ansys also develops software for data management and backup, academic research and teaching. Ansys software is sold on an annual subscription basis.

5.2.2. HISTORY

The first commercial version of Ansys software was labeled version 2.0 and released in 1971. At the time, the software was made up of boxes of punch cards, and the program was typically run overnight to get results the following morning. In 1975, non-linear and thermo-electric features were added. The software was exclusively used on mainframes, until version 3.0 (the second release) was introduced for the VAXstation in 1979. Version 3 had a command line interface like DOS.

In 1980, Apple II was released, allowing Ansys to convert to a graphical user interface in version 4 later that year. Version 4 of the Ansys software was easier to use and added features to simulate electromagnetism. In 1989, Ansys began working with Compuflo. Compuflo's Flotran fluid dynamics software was integrated into Ansys by version 5, which was released in 1993. Performance improvements in version 5.1 shortened processing time two to four-fold and was followed by a series of performance improvements to keep pace with advancements in computing. Ansys also began integrating its software with CAD software, such as Autodesk.

In 1996, Ansys released the Design Space structural analysis software, the LS-DYNA crash and drop test simulation product, and the Ansys Computational Fluid Dynamics (CFD) simulator. Ansys also added parallel processing support for PCs with multiple processors. The educational product Ansys was introduced in 1998. Version 6.0 of the main Ansys product was released in December 2001. Version 6.0 made large-scale modeling practical for the first time, but many users were frustrated by a new blue user interface. The interface was redone a few months later in 6.1. Version 8.0 introduced the Ansys multi-field solver, which allows users to simulate how multiple physics problems would interact with one another.

Version 8.0 was published in 2005 and introduced Ansys' fluid-structure interaction software, which simulates the effect structures and fluids have on one another. Ansys also released its Probabilistic Design System and DesignXplorer software products, which both deal with probabilities and randomness of physical elements. In 2009 version 12 was released with an overhauled second version of Workbench. Ansys also began increasingly consolidating features into the Workbench software.

Version 15 of Ansys was released in 2014. It added some new features for composites, bolted connections, and better mesh tools. In February 2015, version 16 introduced the AIM physics engine and Electronics Desktop, which is for semiconductor design. The following year, version 17 introduced a new user interface and performance improvement for computing fluid dynamics problems. In January 2017, Ansys released version 18. Version 18 allowed users to collect real-world data from products and then incorporate that data into future simulations. The Ansys Application Builder, which allows engineers to build, use, and sell custom engineering tools, was also introduced with version 18.

5.2.3. INTRODUCTION TO ANSYS MECHANICAL (WORKBENCH)

Description:

The ANSYS Workbench environment is an intuitive up-front finite element analysis tool that is used in conjunction with CAD systems and/or Design Modeler. ANSYS Workbench is a software environment for performing structural, thermal, and electromagnetic analyses. The class focuses on geometry creation and optimization, attaching existing geometry, setting up the finite element model, solving, and reviewing results. The class will describe how to use the code as well as basic finite element simulation concepts and results interpretation.

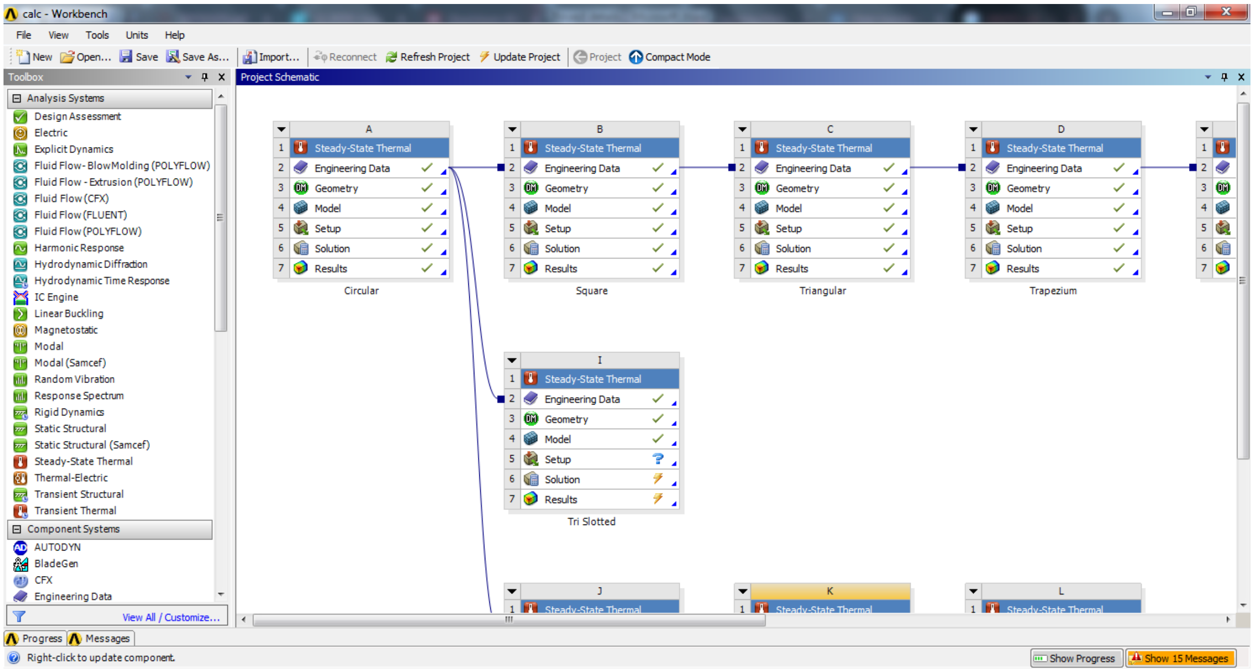


Fig.5.3. ANSYS Workbench window

Topics Include:

- Workbench GUI
- Design Modeler
- Overview of FEA
- Engineering Data
- Meshing
- Parametric Modeling
- Advanced Loads and Boundary Conditions
- Assemblies
- Multiple Load Steps
- Coordinate Systems

- Post processing
- Intro to Thermal Analysis
- Modal Analysis

In this project, predominantly steady state thermal analysis system is used for analyzing the thermal properties of the model. The usage of the software for analysis is explained separately in the upcoming chapters.

CHAPTER-6

NUMERICAL ANALYSIS

6.1. INTRODUCTION

In the numerical analysis we use mathematical approach in calculating the heat transfer rate. The various formulas are obtained from mathematical derivations under defined boundary conditions. We have calculated the value of heat transfer of all sections, namely Circular, Square, Triangular, Trapezoidal and Elliptical with same perimeter, heat transfer co-efficient, thermal conductivity and length and only variation is observed in Cross sectional area. The diameter of circular cross section is assumed (7mm) and the values of perimeter and area is calculated. This value of perimeter for circular section (0.02199m) is kept constant and the sides of other cross sections are derived from it and calculations are done. Al 6082 T6, a widely used material for manufacturing air cooled IC engines with thermal conductivity of 180W/m^{o}C. The ambient temperature is considered as 30^{o}C and heat transfer co-efficient is 12W/m^{2o}C.

6.2. THERMAL PROPERTIES AND BOUNDARY CONDITIONS

Aluminium is one of the most widely used materials for manufacturing of cylinder heads. Among them Al 6082 T6, a recently developed and widely used material for manufacturing air cooled IC engines is used. The thermal properties and boundary conditions of the analysis are shown in table 6.1.

Table.6.1. Boundary conditions and thermal properties

Property	Value
Thermal Conductivity	180 W/m^{o}C
Heat transfer co-efficient	12 W/m^{2o}C
Ambient temperature	30^{o}C

Bore temperature	350°C
Length of fin	30mm

To calculate the heat transfer following assumptions are made:

- There is no change in thermal conductivity with changes in temperature of material.
- The ambient temperature is constant and any changes due to the heat transfer is neglected
- The tip of the fin is considered convecting
- The flow of heat is one dimensional

6.3. FORMULAS INVOLVED AND CALCULATIONS

6.3.1. FORMULAS INVOLVED

$$Q = (T_b - T_\infty) \frac{\tanh(mL) + (h/mK)}{1 + (h/mK)\tanh(mL)} (hPKA)^{0.5}$$

Where, m is a constant $=\sqrt{(hp/KA)}$, m^{-1}

h/mK is biot number

Q is heat transfer, W

T_b is base temperature of fin, °C

T_∞ is ambient temperature, °C

L is length of the fin, m

h is heat transfer co efficient

K is thermal conductivity of Aluminium alloy

A is area of cross section of the fin, m^2

P is perimeter of cross section, m

6.3.2. CALCULATIONS

Circular Cross Section

Diameter of circle = 7mm

$m = 6.172\ m^{-1}$ $mL = 0.18516$ $h/mk = 0.0108$

$P = 0.02199m$ $A = 3.848 \times 10^{-5}\ m^2$

$$Q = (347-30)\ \frac{\tanh(0.18516) + (0.0108)}{1 + 0.0108 \tanh(0.18516)}\ (0.04093) = 2.510\ W$$

$Q = 65221.04\ W/m^2$

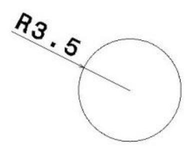


Fig.6.1. Circular cross section

Square Cross Section

Side of the square = 5.4975mm

$m = 6.9647\ m^{-1}$ $mL = 0.20894$ $h/mk = 9.572 \times 10^{-3}$

$P = 0.02199m$ $A = 3.0225 \times 10^{-5}\ m^2$

$$Q = (347-30)\ \frac{\tanh(0.20894) + (9.572 \times 10^{-3})}{1 + (9.572 \times 10^{-3}) \tanh(0.20894)}\ (0.03789) = 2.584\ W$$

$Q = 85485.76\ W/m^2$

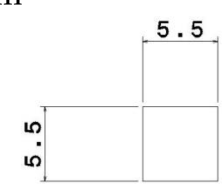


Fig.6.2. Square cross section

Triangular Cross section

Side of the equilateral triangle = 7.33mm

$m = 7.938\ m^{-1}$ $mL = 0.23814$ $h/mk = 8.3984 \times 10^{-3}$

$P = 0.02199m$ $A = 2.32653 \times 10^{-5}\ m^2$

$$Q = (347-30)\ \frac{\tanh(0.23814) + (8.3984 \times 10^{-3})}{1+(8.3984 \times 10^{-3})\tanh(0.23814)}\ (0.03324) = 2.546\ W$$

$Q = 1.09432 \times 10^5\ W/m^2$

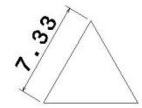


Fig.6.3. Triangular cross section

Trapezoidal Cross Section

Length of base of trapezoid = 8.796 mm

Length of top and sides surfaces = 4.398 mm

$m = 7.6384\ m^{-1}$ $mL = 0.22915$ $h/mk = 8.7278 \times 10^{-3}$

$P = 0.02199m$ $A = 2.51265 \times 10^{-5}\ m^2$

$$Q = (347-30)\ \frac{\tanh(0.22915) + (8.7278 \times 10^{-3})}{1+ (8.7278 \times 10^{-3})\tanh(0.22915)}\ (0.034547) = 2.557\ W$$

$Q = 1.01767 \times 10^5\ W/m^2$

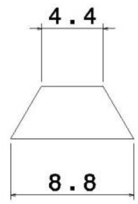


Fig.6.4. Trapezoidal cross section

Elliptical Cross Section

Length of major axis = 4.42695 mm

Length of minor axis = 2.21348 mm

$m = 6.9008\ m^{-1}$ $mL = 0.207$ $h/mk = 9.6607 \times 10^{-3}$

$P = 0.02199m$ $A = 3.07844 \times 10^{-5} m^2$

$$Q = (347-30) \frac{\tanh(0.207) + (9.6607 \times 10^{-3})}{1 + (9.6607 \times 10^{-3})\tanh(0.207)} (0.03824) = 2.586\ W$$

$Q = 84003\ W / m^2$

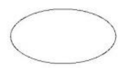


Fig.6.5. Elliptical cross section

The above calculations can be summarized in the table.6.2.

Table.6.2. Results of Heat transfer of regular shape

Cross sectional area of Fin	**Heat Transfer W/m^2**
Circular	65221
Square	85485
Triangle	109452
Trapezium	101767
Ellipse	84003

6.4. OPTIMIZED MODEL

From the calculations it is very clear that the value of heat transfer rate is the highest for triangular cross section. It should also be noted that the triangular fin is the fin with lowest cross section area. Thus it further strengthens the postulate that thinner the fin, more the heat transfer rate. Since decreasing the cross sectional area has a positive effect on increasing the heat transfer flux, further the cross section of triangular fin is reduced and heat transfer flux is calculated. The Optimized designs are:

- Triangular Cross Section with a slot

- Triangular Cross section with Rectangular grooves on sides

Triangular Cross Section with a slot

A slot is made centrally in the fin in the axial direction as shown. The dimensions of the slot are 0.5 times the dimensions of triangular fin section. Hence the fin is made thinner and it also allows more surface area for convection heat transfer to take place. Then the calculations are made for finding out the heat transfer rate.

Side of equilateral triangle = 7.33 mm

Side of slot (equilateral triangle) = 3.665 mm

Surface area = 9.8955 mm^2

$m = 11.226\ m^{-1}$ $\quad mL = 0.33678$ $\quad h/mk = 5.939 \times 10^{-3}$

$P = 0.032985m$ $\quad A = 1.7449 \times 10^{-5} m^2$

$$Q = (347-30)\ \frac{\tanh(0.33678) + (5.939 \times 10^{-3})}{1 + (5.939 \times 10^{-3})\tanh(0.33678)}\ (0.0353) = 3.692\ W$$

$Q = 211569\ W/m^2$

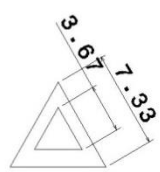


Fig.6.6. Triangular cross section with a slot of scale 0.5

Triangular Cross section with Rectangular grooves on sides

Grooves of rectangular shape are made on the edges of the triangle to make the fin thinner as shown. Then the calculations are made for finding out the heat transfer rate as shown below

Length of side of equilateral triangle = 7.33 mm

Width of groove = 4 mm

Depth of groove = 0.5 mm

Surface area = 7.497 mm^2

$m = 9.82\ m^{-1}$ $mL = 0.2947$ $h/mk = 6.7889 \times 10^{-3}$

$P = 0.02499m$ $A = 1.7265 \times 10^{-5} m^2$

$$Q = (347-30) \frac{\tanh(0.2947) + (6.7889 \times 10^{-3})}{1 + (6.7889 \times 10^{-3})\tanh(0.2947)} (0.0305) = 3.348\ W$$

$$Q = 193897\ W/m^2$$

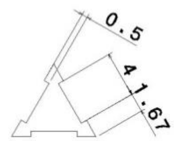


Fig.6.7. Triangular cross section with a Rectangular groove

The results from the optimized design are summarized in the table.6.3.

Table.6.3. Results of heat transfer calculations of optimized design

Shape	Heat Transfer W/m^2
Triangular Cross Section with a slot	211569
Triangular Cross section with Rectangular grooves on sides	193897

The graph of heat transfer rates of all the sections is plotted for analyzing the results of the investigation as shown in the fig.6.8.

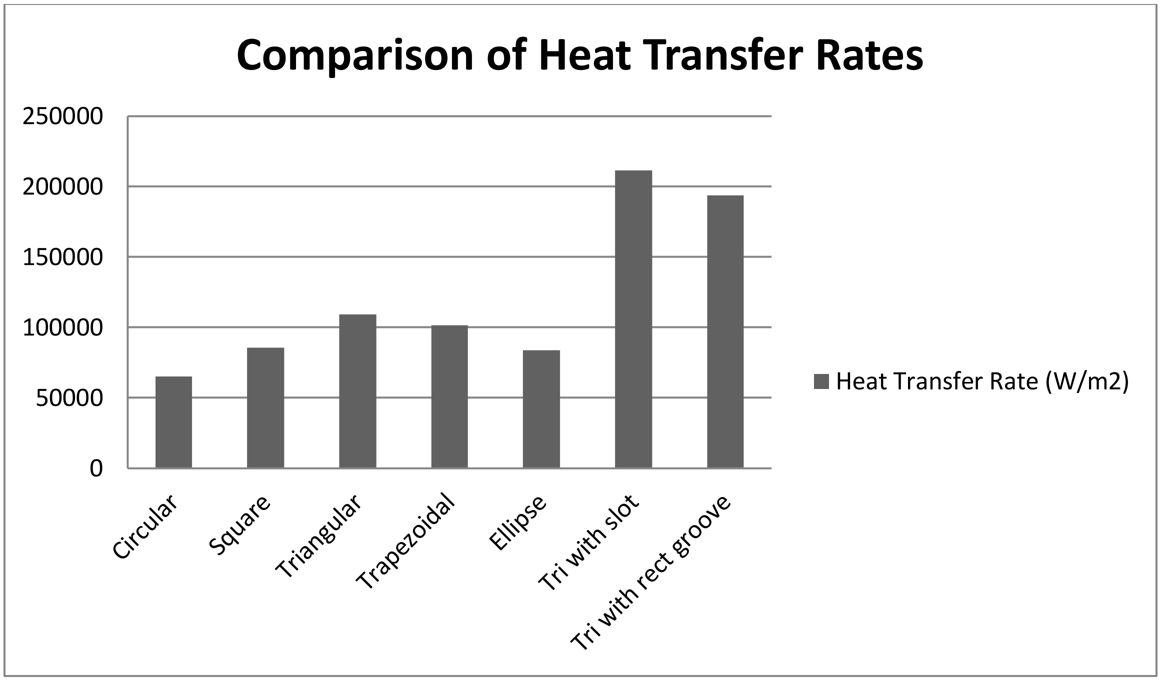


Fig.6.8. Comparison of the heat transfer rates of various sections

From the fig.6.8, it is very clear that the heat transfer rate has considerably increased with the introduction of the optimized design. This increase is attributed to the decrease in cross sectional area and increase in surface area of the fin which provides more area for convective heat transfer to take place. Among the optimized design, the fin with slot has higher heat transfer although the cross sectional areas of the two fins is almost equal. Here the surface area plays a role in increasing the heat transfer.

The decrease in cross section also suggests that an equivalent decrease in the material used and thus reduction in weight of the cylinder. This reduction in weight also improves the weight to power ratio and thus enhances the efficient performance of the engine. The percent increase in heat transfer rate from that of the regular triangular fin is shown in the following table.6.4.

Table.6.4. Increase in heat transfer with fins of equal length

Shape	% Increase
Triangular Cross Section with a slot	93.3
Triangular Cross section with Rectangular grooves on sides	77.2

Further a small proportion of material can be saved if the optimized design is adopted in the engine cylinder. This is analyzed by calculating the volume of the cylinder and multiplying it with the density. It is demonstrated using ANSYS as shown in next chapter. Moreover, the calculations made here are also verified in next chapter.

CHAPTER-7

FINITE ELEMENTAL ANALYSIS USING ANSYS

7.1. INTRODUCTION

The verification of the results obtained in numerical method is done through analysis in ANSYS. ANSYS Workbench is a powerful multidisciplinary software package that allows the user to perform complex analysis like electromagnetic, thermal, structural, etc….. . Although modeling can be performed in ANSYS considering the complexity of the design, CATIA V5 is used and for meshing and analysis, ANSYS is preferred. The usage of software is separately explained below.

7.2. MODELING

CATIA V5 is a modeling and drafting software. As mentioned above, the complexity of designing the engine with the fins is the reason for using the software. Part Modeling System in CATIA V5 is used here. First the cylinder is modeled. The XY Plane is selected for sketching. Centre Rectangle command is used and chamfer is applied at the corners. Then the sketch window is exited and the pad is applied to obtain the cylinder of length 120mm.

For Bore, the top face of cylinder is selected and circle is sketched of required diameter (say 40mm). The Pocket Command is used to remove material for the bore. The other aspects of design are made as shown in the Fig.5.1.

For the Fin Profile, different geometries of fin are drawn on the face perpendicular to the face of cylinder and extruded to 30mm. This extruded fin is then subjected to the circular pattern command to obtain a total of 96 fins on the cylinder surface. The different geometries used are circle, square, triangle, ellipse and trapezium as shown in figs.7.2-7.6. The dimensions of the fins are same as that of the respective ones discussed in chapter-6. The files of different profiles are saved in the .igs format for import into ANSYS Workbench.

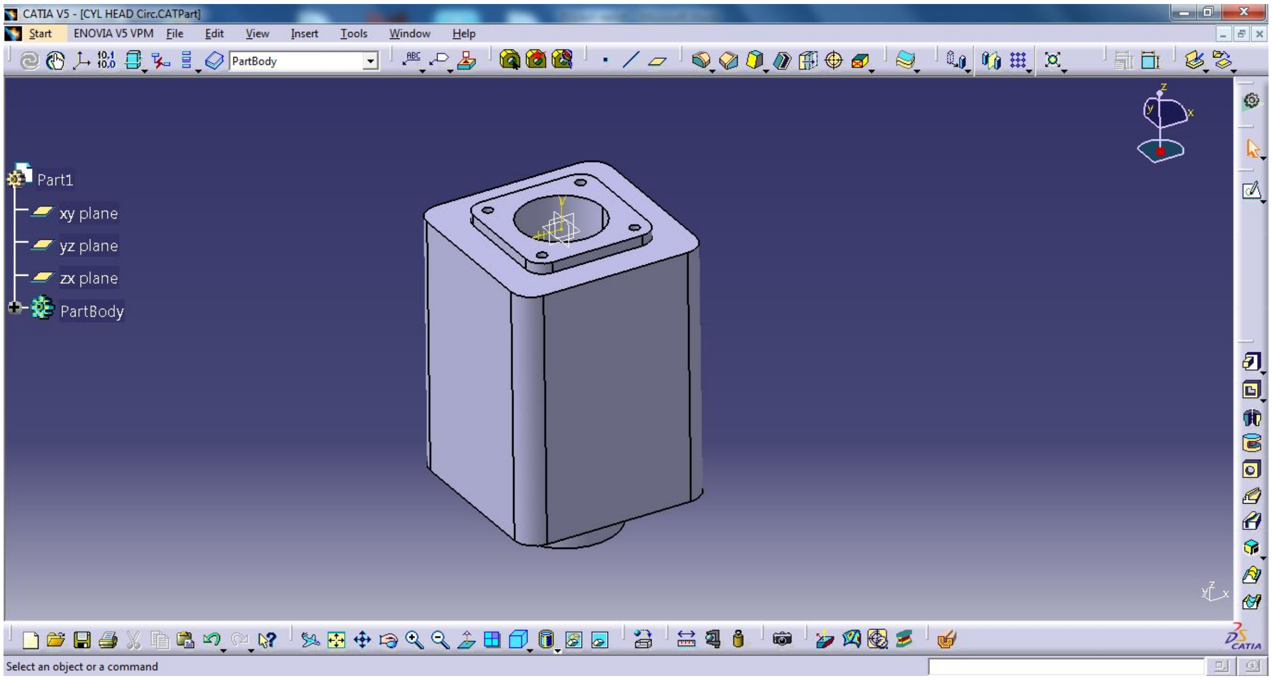
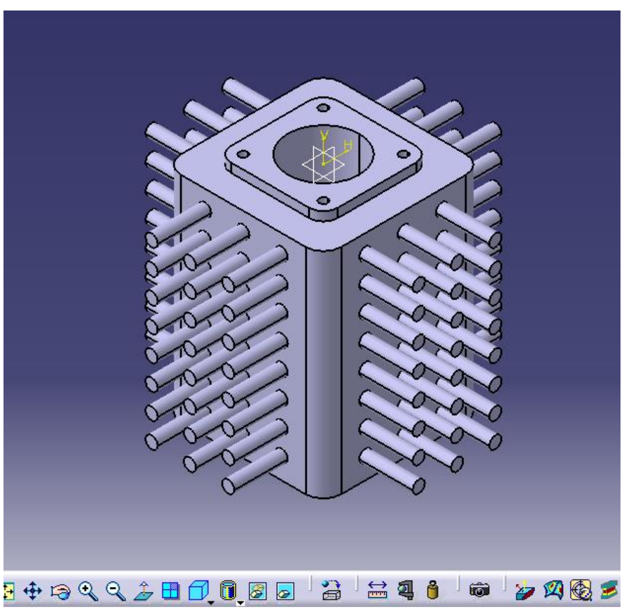


Fig.7.1. Model of cylinder done in CATIA V5



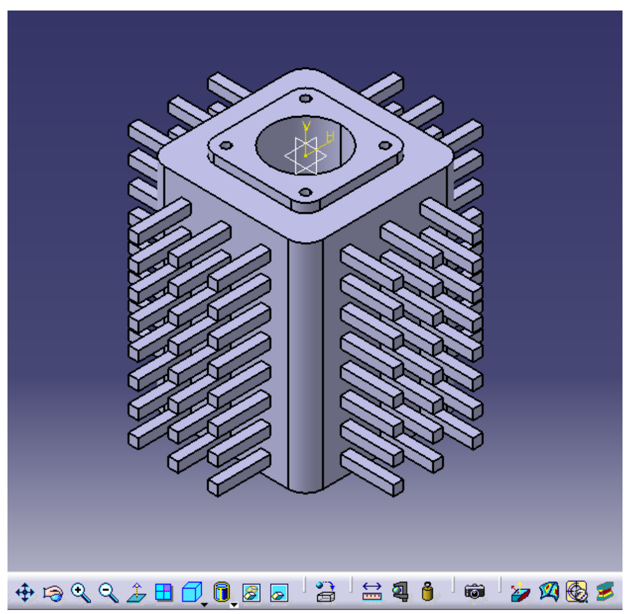
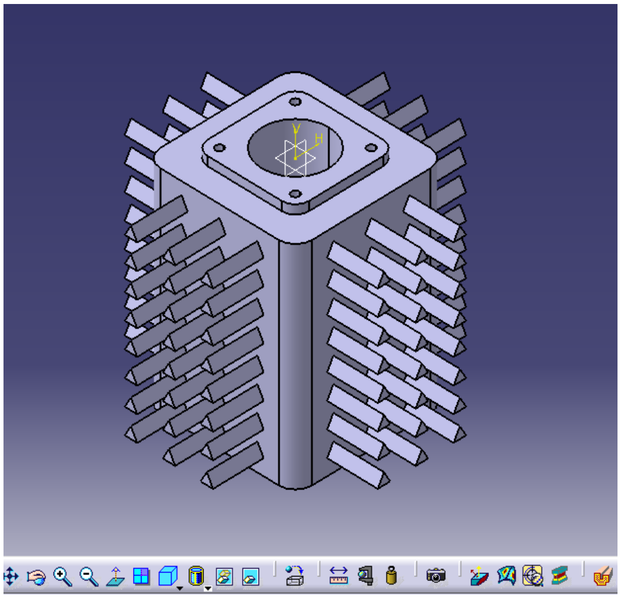


Fig.7.2. Circular cross section **Fig.7.3. Square cross section**



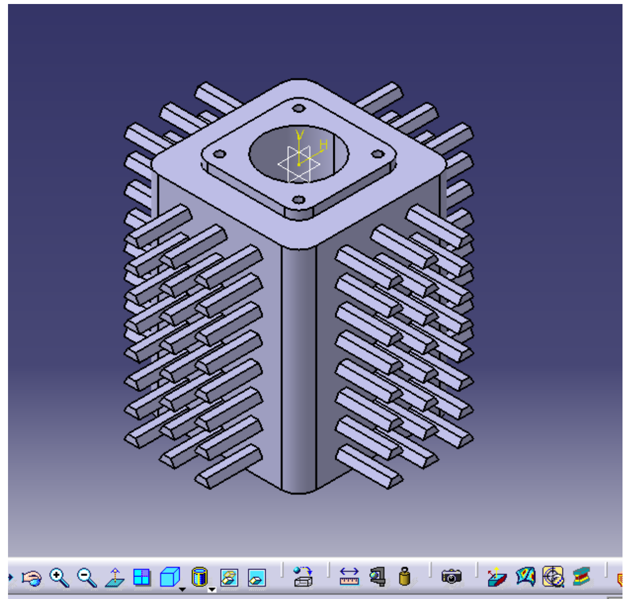


Fig.7.4. Triangular cross section **Fig.7.5. Trapezoidal cross section**

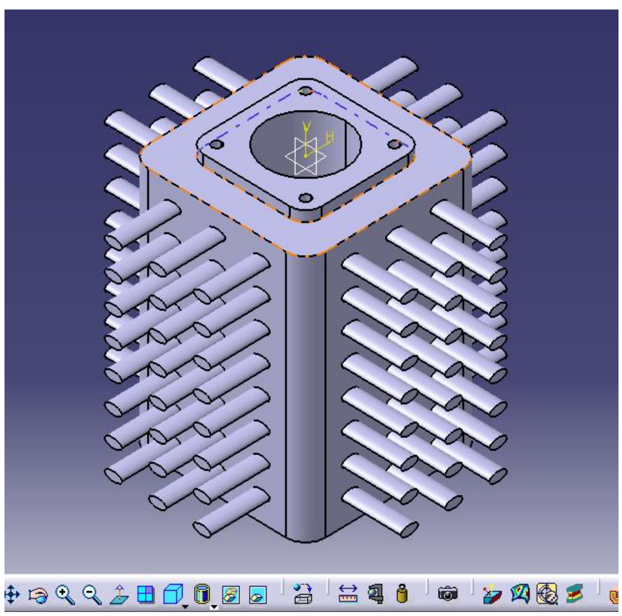


Fig.7.6. Elliptical cross section

7.3. STEADY STATE THERMAL ANALYSIS

Since the fin operates in steady state conditions, Steady state thermal analysis system is selected. The various operations following it are explained below.

7.3.1. ENGINEERING DATA UPDATION

The engineering data updating component system is selected to update the material and thermal properties. The default material is always structural steel. Since our required material is Al 6082 and it is not available by default, we have to assign a new material. Click “add new material” and name it as Al 6082. The value of thermal conductivity is entered as 180 W/m°C as shown in fig.7.7.

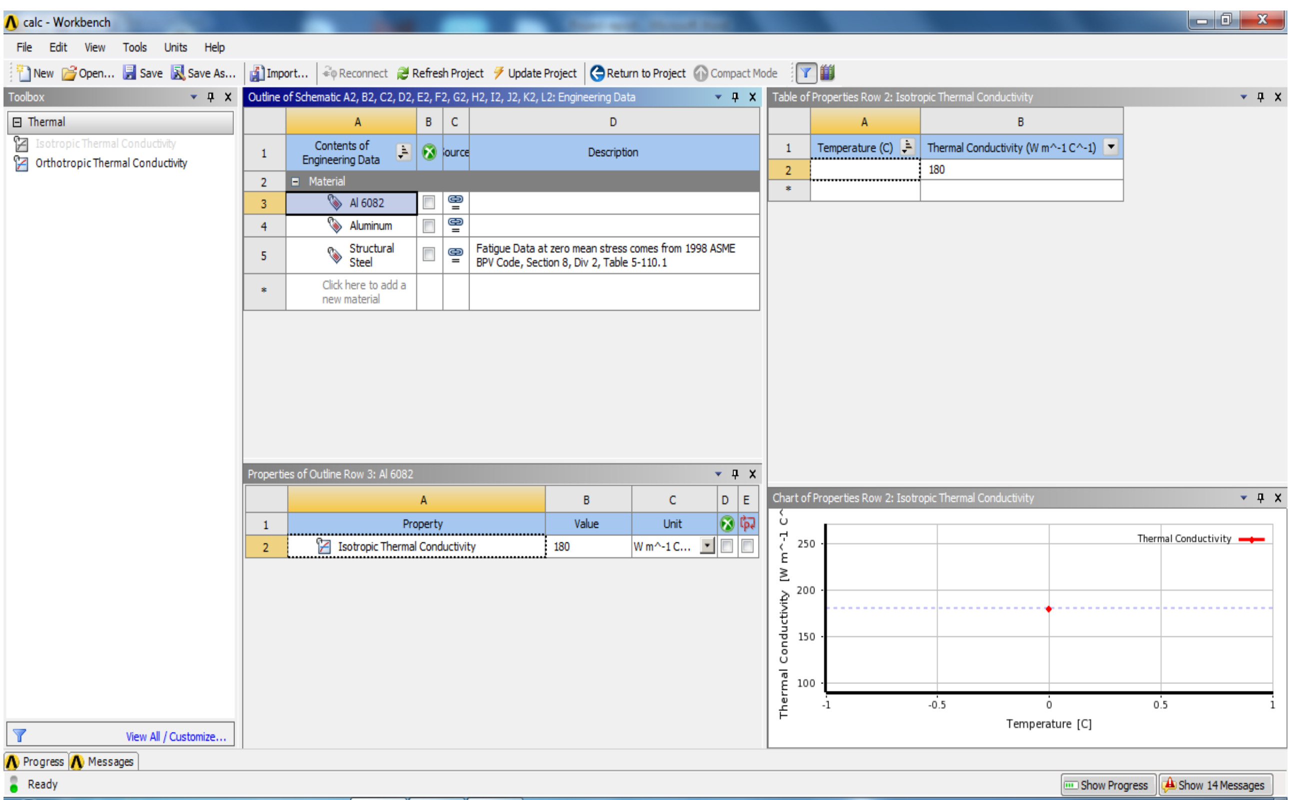


Fig.7.7. Engineering Data component system

7.3.2. GEOMETRY MODELER

Now the project is updated and returned to the workbench window. Now Geometry component system is opened and the previously saved model in .igs format is opened and updated. Exit the Geometry component system. Design Modeler for circular profile is shown in the fig.7.8.

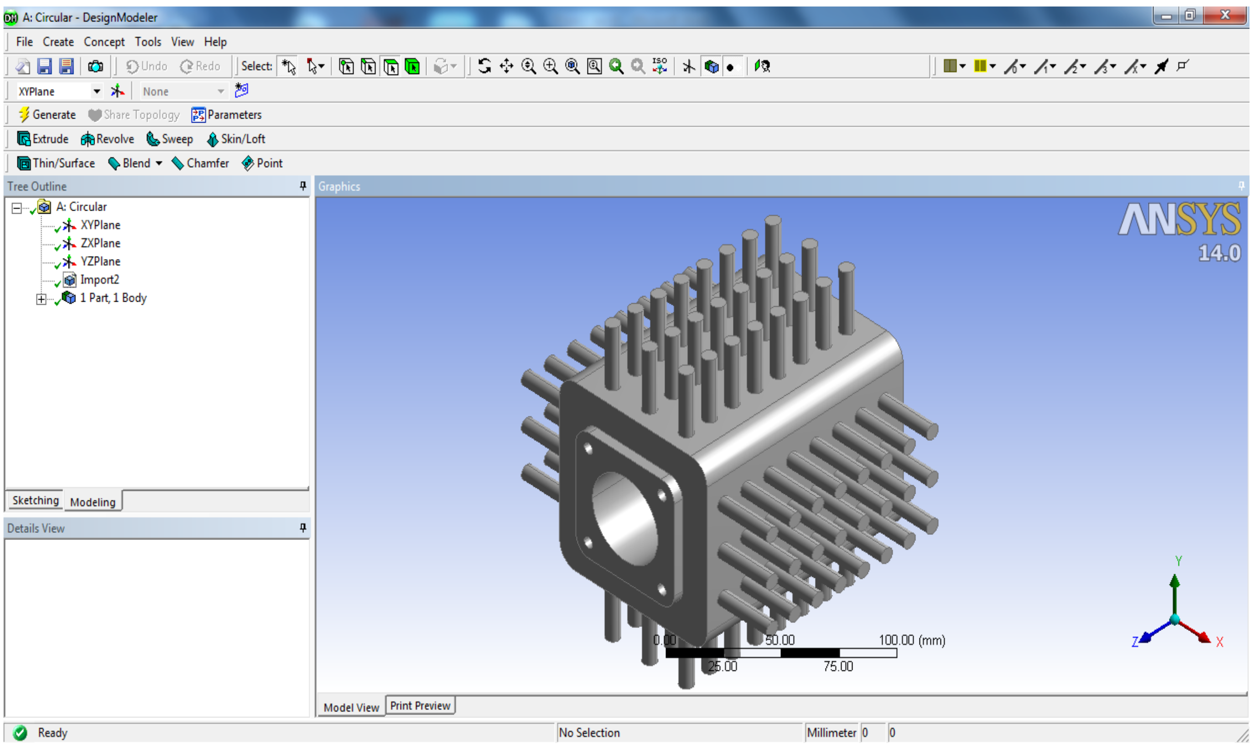


Fig.7.8. Geometry modeler updating for Circular profile

7.3.3. MESHING

The Mechanical model for the analysis system is opened. In the outline tree, the solid sub tree is selected and the assigned material is set as Al 6082. Next the meshing sub tree is opened and the parameters for meshing are set. The Physics preference is Mechanical and relevance is 0. Triangular mesh is used and the use of advanced sizing feature is turned off since the sizes of the triangular mesh elements are much smaller than the filleted edges. The Element size is default and the relevance center is fine. The smoothening is medium, transition is fast and inflation option is smooth transition. The mesh for Circular profile is shown in the fig.7.9. The no of nodes and elements for each profile is shown in table 7.1.

Table 7.1. Details of mesh

Shape	Nodes	Elements
Circle	50259	27352
Square	57419	31111
Triangle	44310	23470
Trapezium	61737	33627
Ellipse	55795	29868

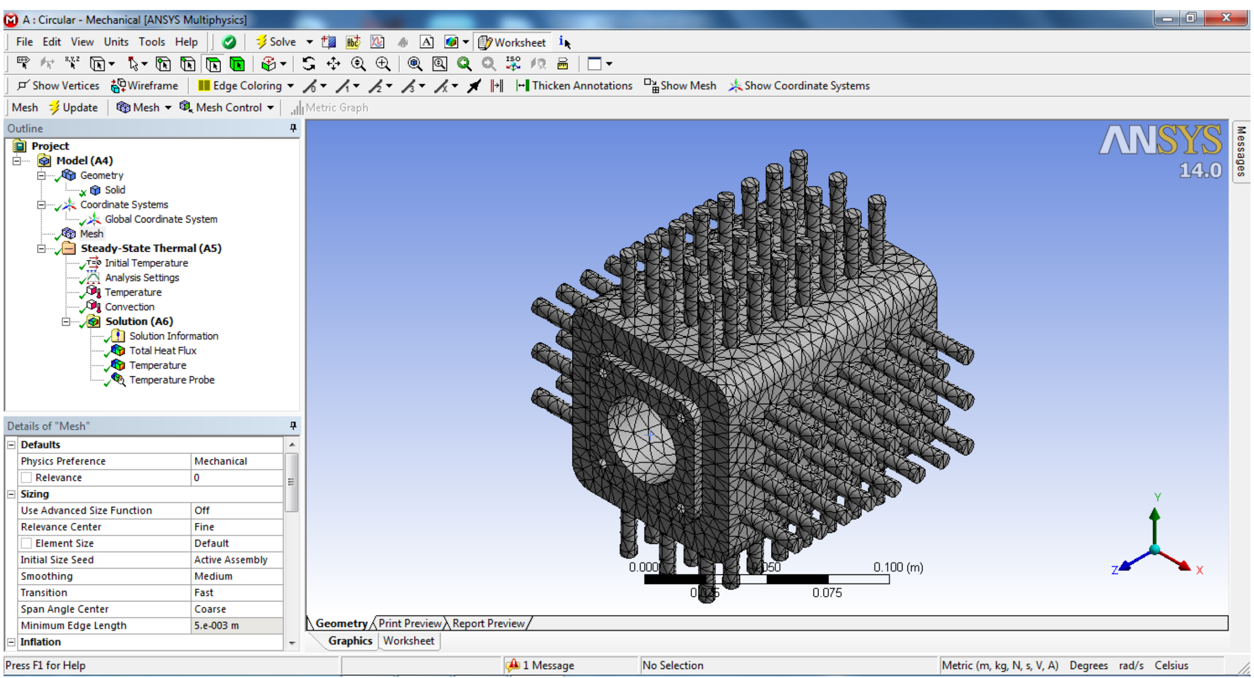


Fig.7.9. Meshing of Circular profile

7.3.4. BOUNDARY CONDITIONS

To perform the steady state thermal analysis, the boundary conditions have to be set. This is done through the steady state thermal sub system in the tree. The required boundary conditions are same as those discussed in chapter-6 and mentioned in table 7.2.

Table 7.2. Boundary Conditions

Boundary Condition	Value
Bore temperature	350 ^{o}C
Convection Co-efficient	12W/m^{2o}C
Ambient temperature	30 ^{o}C

The geometry selection of Circular profile for applying the boundary conditions is specified in fig.7.10. In a similar way, the geometries for other profiles are made.

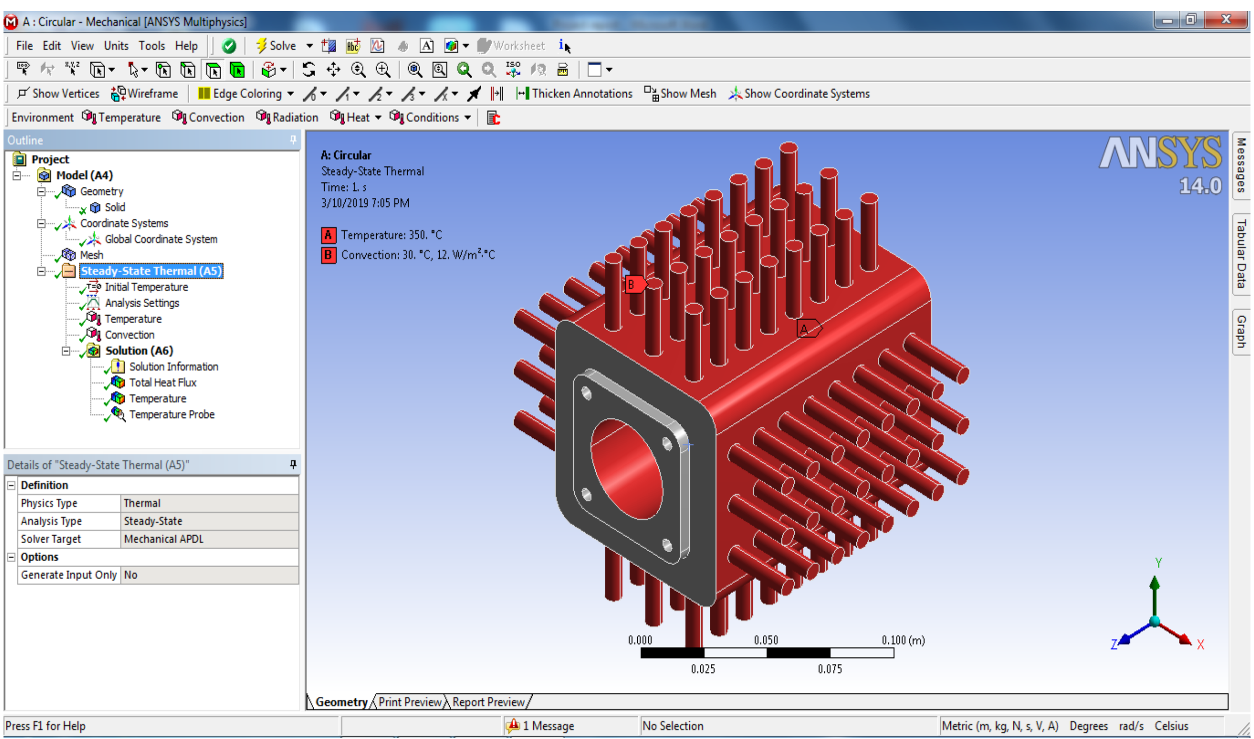


Fig.7.10. Geometry selection for applying boundary conditions

7.4. RESULTS OF ANALYSIS

For the result part, thermal flux is selected for heat transfer analysis and temperature is selected for finding the temperature at tip of the fin. Temperature at tip is a direct indicator of heat lost by convection. Lesser the temperature at tip, more heat transfer has taken place and thus the cylinder is cooled more. The results for all the sections are obtained and are shown in the fig.7.11– 7.20.

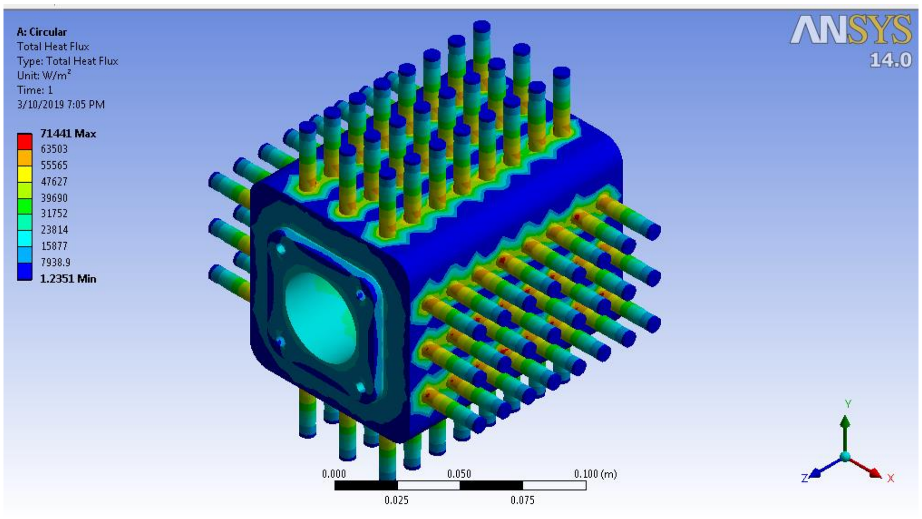


Fig.7.11. Heat transfer in Circular profile

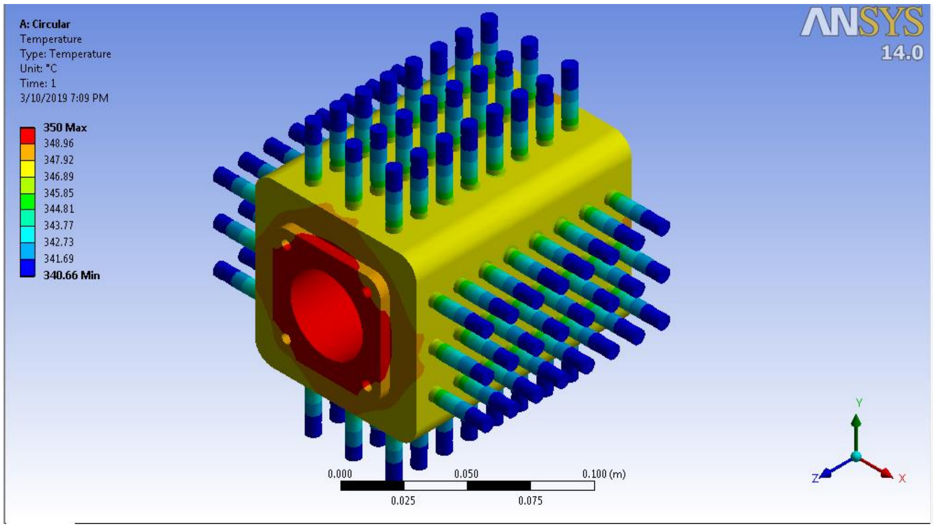


Fig.7.12. Temperature distribution in circular profile

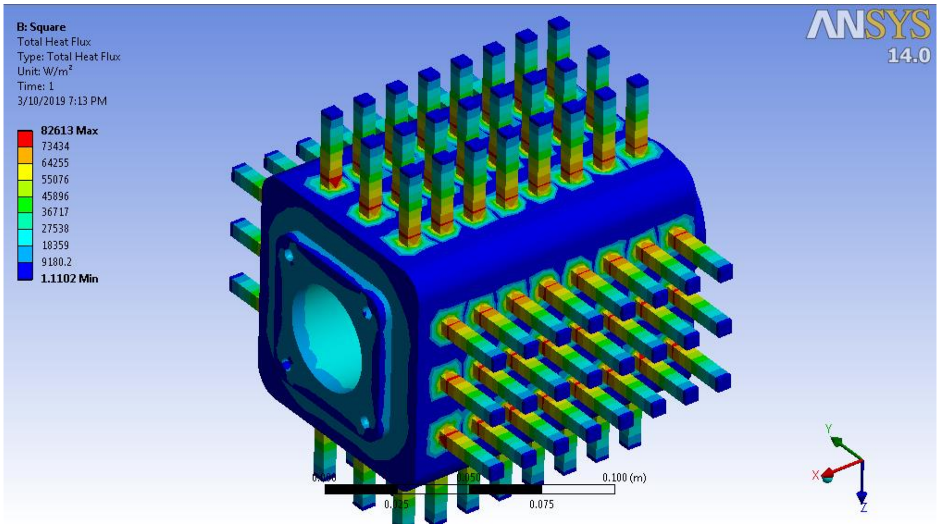


Fig.7.13. Heat transfer in Square profile

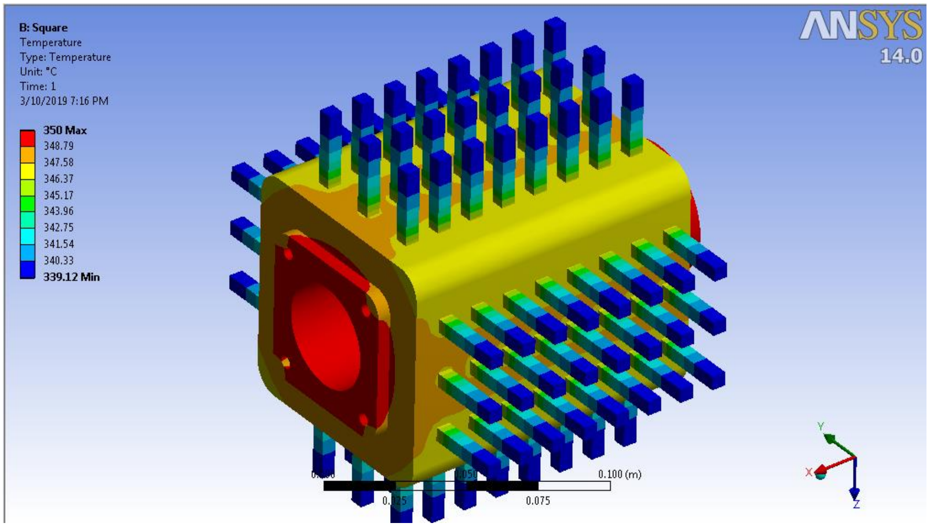


Fig.7.14.Temperature distribution in square profile

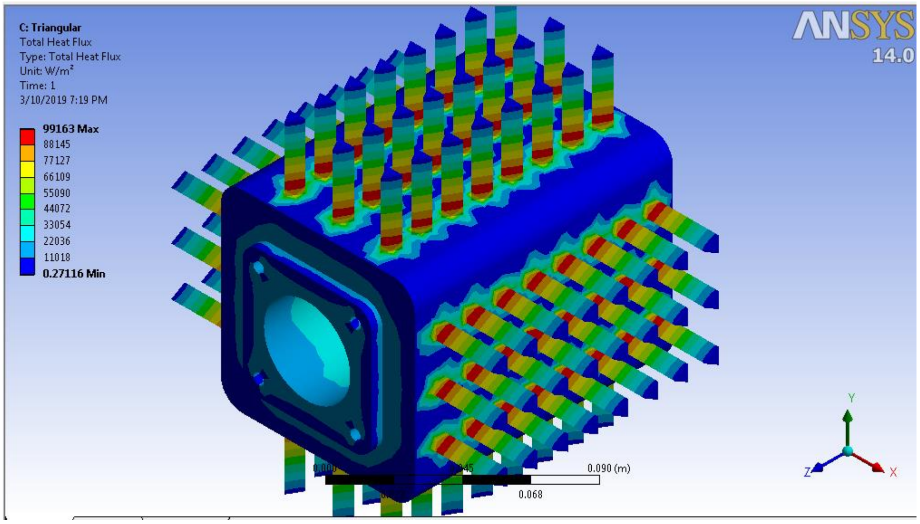


Fig.7.15. Heat transfer in Triangular profile

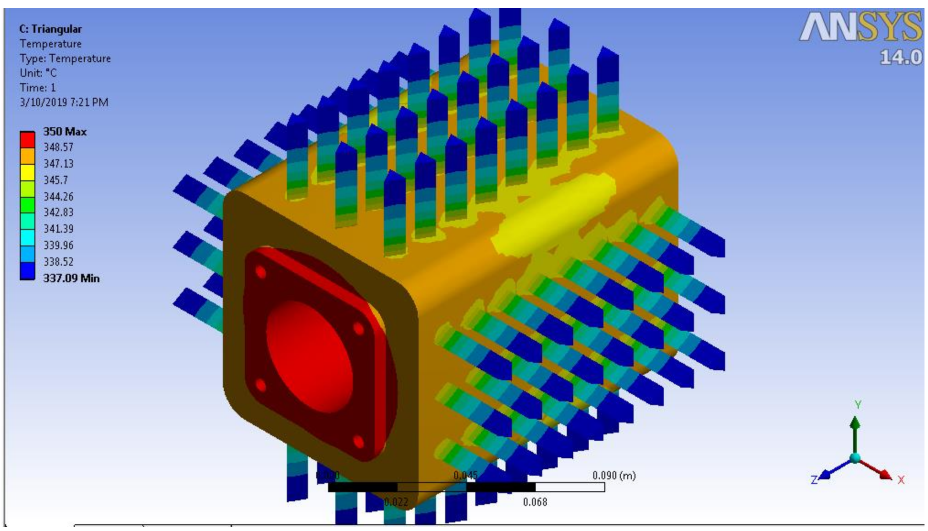


Fig.7.16.Temperature distribution in triangle profile

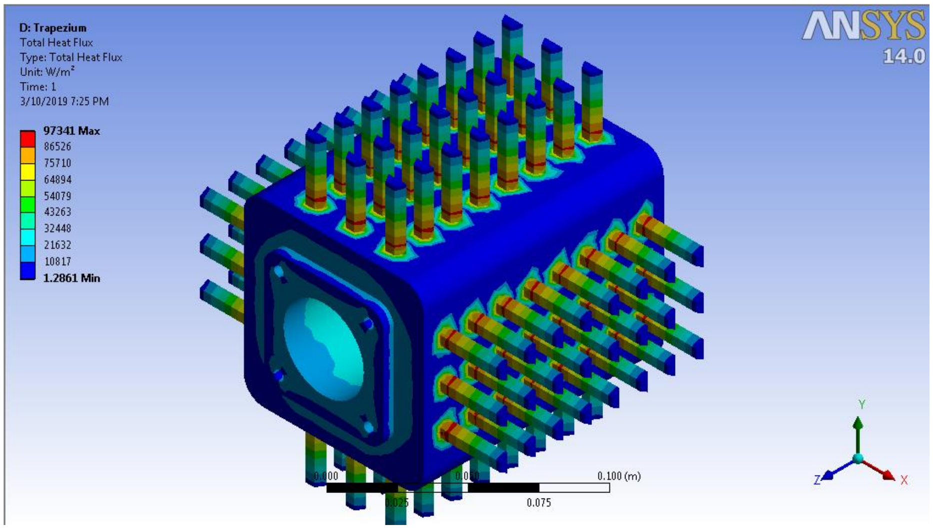


Fig.7.17.Heat transfer in trapezium profile

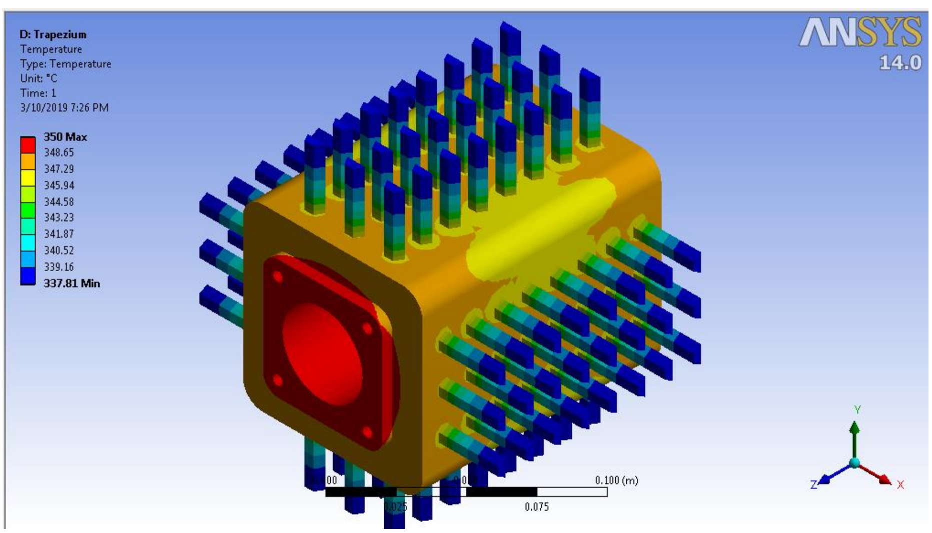


Fig.7.18.Temperature distribution in trapezium profile

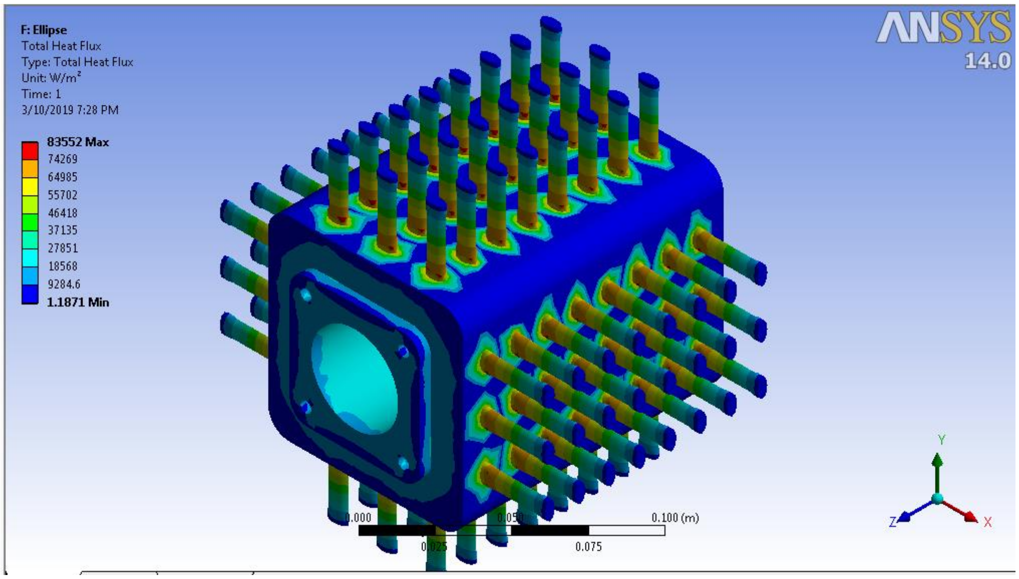


Fig.7.19.Heat transfer in elliptical profile

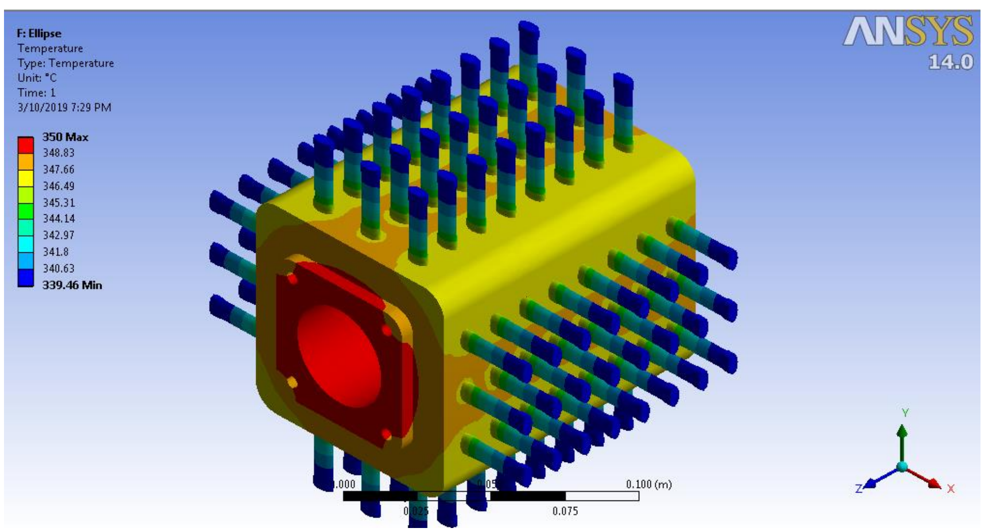


Fig.7.20.Temperature distribution in elliptical profile

From the table 7.3., it is clear that lesser the cross sectional area (A_c), higher the heat transfer rate. The temperature at the end of the fin is also the least for triangular profile fin, indicating that more heat energy is lost in convection. As a result fin with triangular cross section is having higher rate of heat transfer. This is also in accordance with the calculations made in the numerical analysis made in chapter-6. Hence for further increasing the heat transfer rate, the triangular fin is further made thin by making a slot of scale 0.5 profile and rectangular groove. Thus convective surface area is increased for better heat transfer rate.

Table 7.3. Results of finite elemental analysis

ANSYS Analysis		
Shape	**Heat Transfer W/m²**	**Temp at X=30mm (°C)**
Circular	71441	340.66
Square	82613	339.12
Triangle	99163	337.09
Trapezium	97341	337.81
Ellipse	83552	339.46

7.5. OPTIMIZED DESIGN

As mentioned above, the optimized design is made for the triangular fin to increase the heat transfer rate. The process involved in analysis of the optimized fin is same as that of those in regular fins. First the model is created as shown in fig.7.21.-7.22. The dimensions of the fin are same as that of the ones discussed in chapter-6.

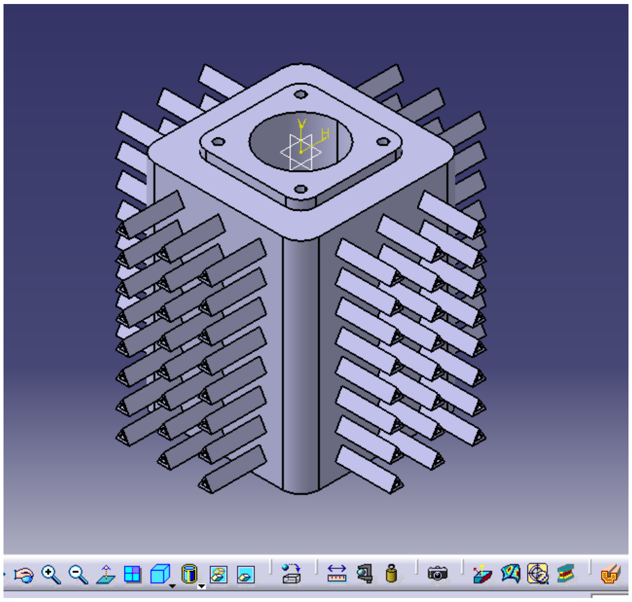


Fig.7.21. Triangular cross section with slot of scale 0.5

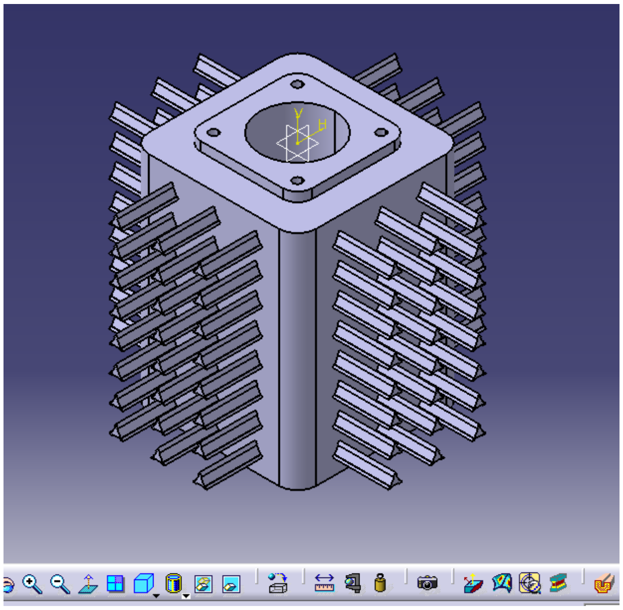


Fig.7.22. Triangular cross section with rectangular groove

The file is saved in the .igs format and imported in ANSYS for analysis. The same engineering data are assigned (i.e.) Al 6082 with thermal conductivity of 180 W/m°C. The model is then generated in the design modeler.

In steady state thermal analysis component system, meshing is done. The mesh preferences are same as that done for regular fins. Further the mesh details for optimized fins are shown in table 7.4. It can be seen that the no of nodes and elements have increased. This is due to the increased surface area of the fin.

Table 7.4. Mesh details for Optimized fins

Shape	Nodes	Elements
Slotted	82901	47423
Rectangle grooved	265380	145409

The steady state sub tree is selected for applying the load. The boundary conditions are same and as mentioned below in table 7.5.

Table 7.5. Boundary Conditions for optimized fins

Boundary Condition	Value
Bore temperature	350 ºC
Convection Co-efficient	12W/m^{2}ºC
Ambient temperature	30 ºC

The geometry selection for the optimized design is same as that of the regular shaped fin and is shown in fig.7.23. Temperature is applied in the bore. The ambient temperature is assumed to be 30 ºC. The convection is applied on the outer surface of the cylinder where the coolant (air) is in contact with it.

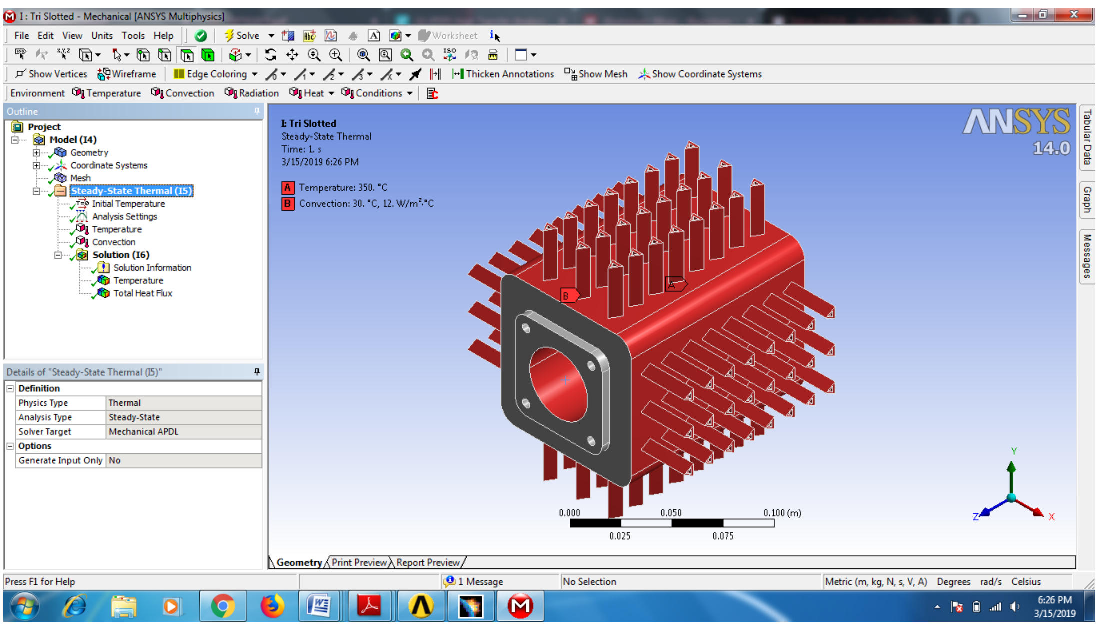


Fig.7.23. Geometry selection of optimized fin for applying load

7.6. RESULTS OF ANALYSIS OF OPTIMIZED DESIGN

The cylinder containing the optimized design is set up as mentioned above. The thermal flux and temperature distribution are obtained as results from the analysis. The results of analysis are shown in the figs.7.24-7.29.

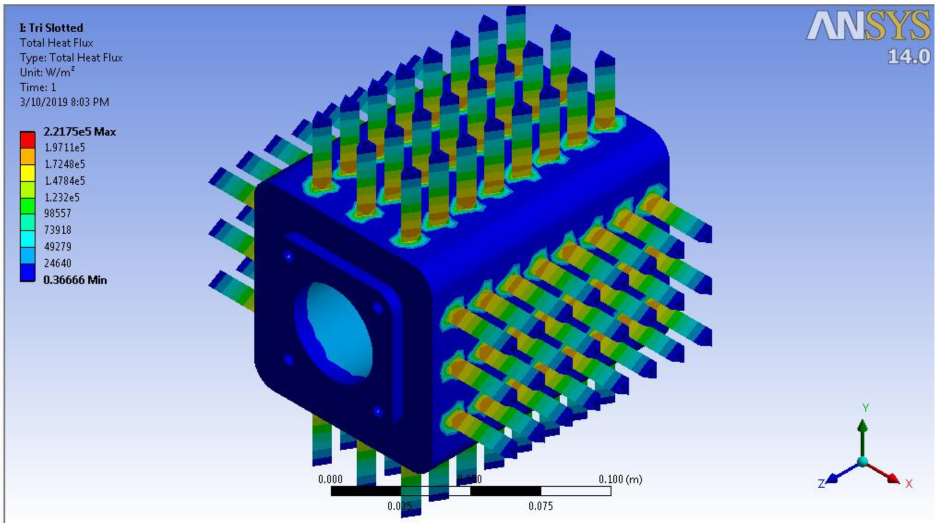


Fig.7.24. Heat transfer for slotted fins

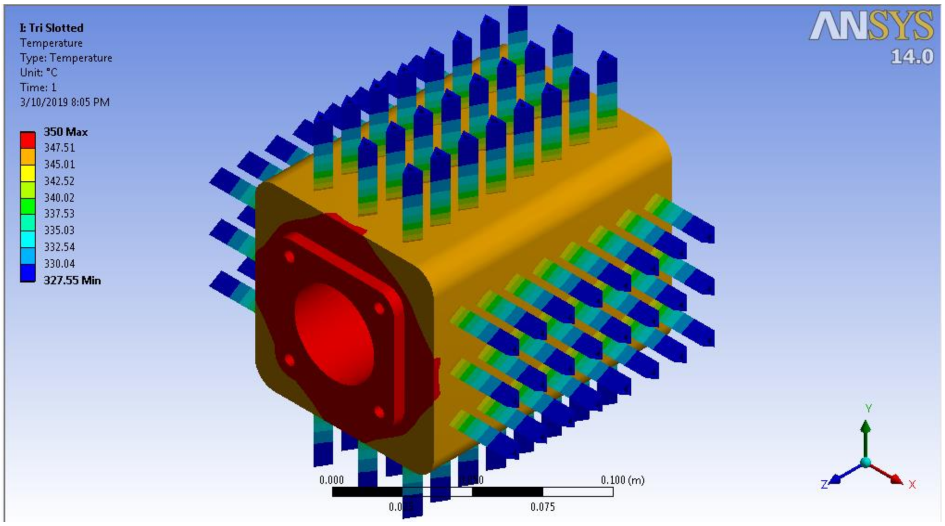


Fig.7.25. Temperature distribution for slotted fins

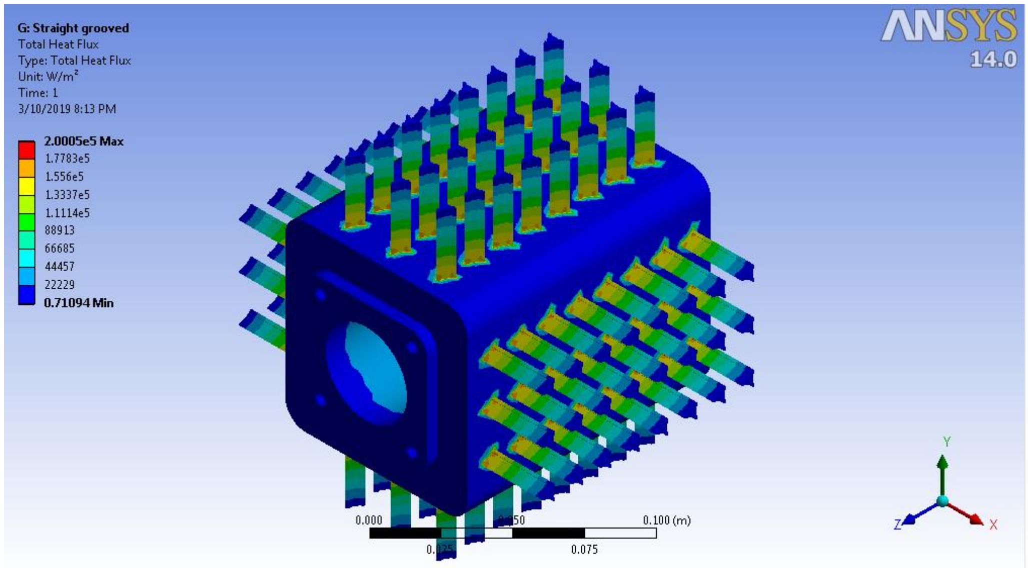


Fig.7.26. Heat transfer for rectangular grooved fins

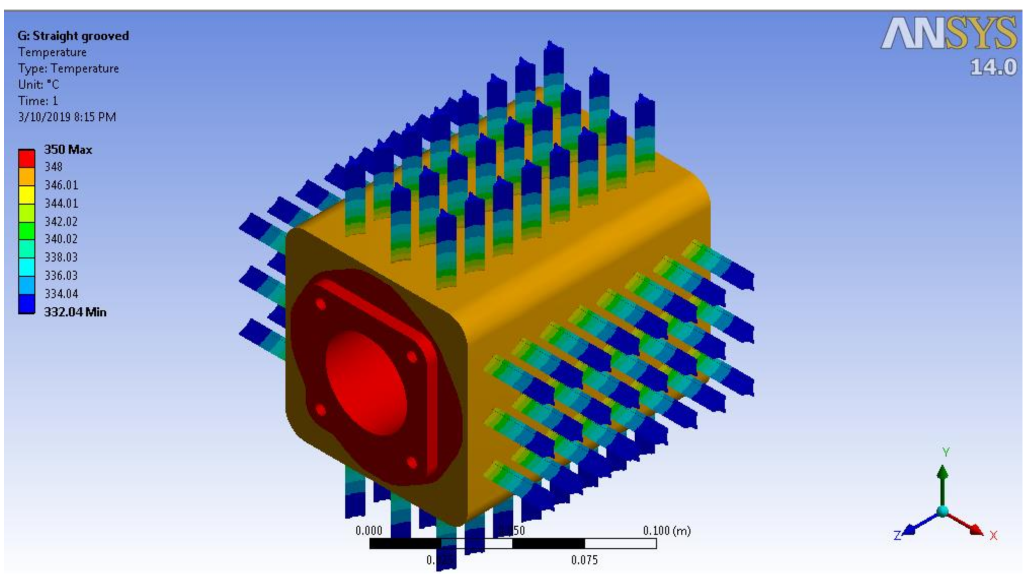


Fig.7.27. Temperature distribution for rectangular grooved fins

The results of the analysis are summarized in the table 7.6 as shown below.

Table 7.6. Results of optimized fin analysis

ANSYS Analysis		
Shape	**Heat Transfer W/m^2**	**Temp at X=30mm (ºC)**
Slotted	221750	327.55
Rectangular groove	200050	331.55

From the analysis results, it is clear that the heat transfer rate has increased considerably. This strengthens the claim and calculations made in the chapter-6. The increase in heat transferred and the decrease in temperature at the tip from the optimized design compared to the regular triangular fin is shown in table 7.7.

Table 7.7. Percent change in heat transfer and the temperature at the tip

Shape	% Increase in heat transfer	% decrease in temperature at X=30mm
Triangular Cross Section with a slot	93.3	2.83
Triangular Cross section with Rectangular grooves on sides	49.8	1.50

Further it is also noted that, a considerable amount of material can be saved by the slot made and the grooves done on the sides. For comparison, the weight of the cylinder is calculated. This weight is calculated by the volume of the cylinder obtained in the steady state analysis component system. The volume of the cylinder is obtained in geometry sub tree under the properties. This volume is obtained by multiplying with the density. The mass of cylinder can be obtained directly by entering the density of Al 6082 directly in the engineering data. This makes the mass of the cylinder to be displayed under properties in geometry sub tree. The mass of the cylinder for comparison is summarized in the table 7.8. The cross section of the fin is also displayed.

The data in the table 7.8 clearly shows a decrease in the mass of the cylinder for the optimized fin in comparison with the regular fin as claimed.

Table 7.8. Mass of cylinder having different profiles

Shape	Cross section Area(A_c) (m^2)	Volume (m^3)	Mass (Kg)
CIRCLE	3.848×10^{-5}	9.3754×10^{-4}	2.53
SQUARE	3.022×10^{-5}	9.1381×10^{-4}	2.47
TRIANGLE	2.327×10^{-5}	8.9370×10^{-4}	2.41
TRAPEZIUM	2.513×10^{-5}	8.9906×10^{-4}	2.43
ELLIPSE	3.078×10^{-5}	9.1517×10^{-4}	2.47
TRI SLOTTED	1.745×10^{-5}	8.6019×10^{-4}	2.30
TRI RECT GROV	1.727×10^{-5}	8.6956×10^{-4}	2.33

CHAPTER-8

CONCLUSION

From the above discussion, we conclude that heat transfer rate has increased for the optimized design. This is illustrated using the comparison shown in fig. 8.1. This increased heat transfer is due to the increased surface area which allows more surfaces for convection to take place and decreased cross sectional area which allows more heat to be conducted per unit area. The increased surface for slotted fin is the interior of the slot which is marginally greater than the groove made in the other case of the optimized model.

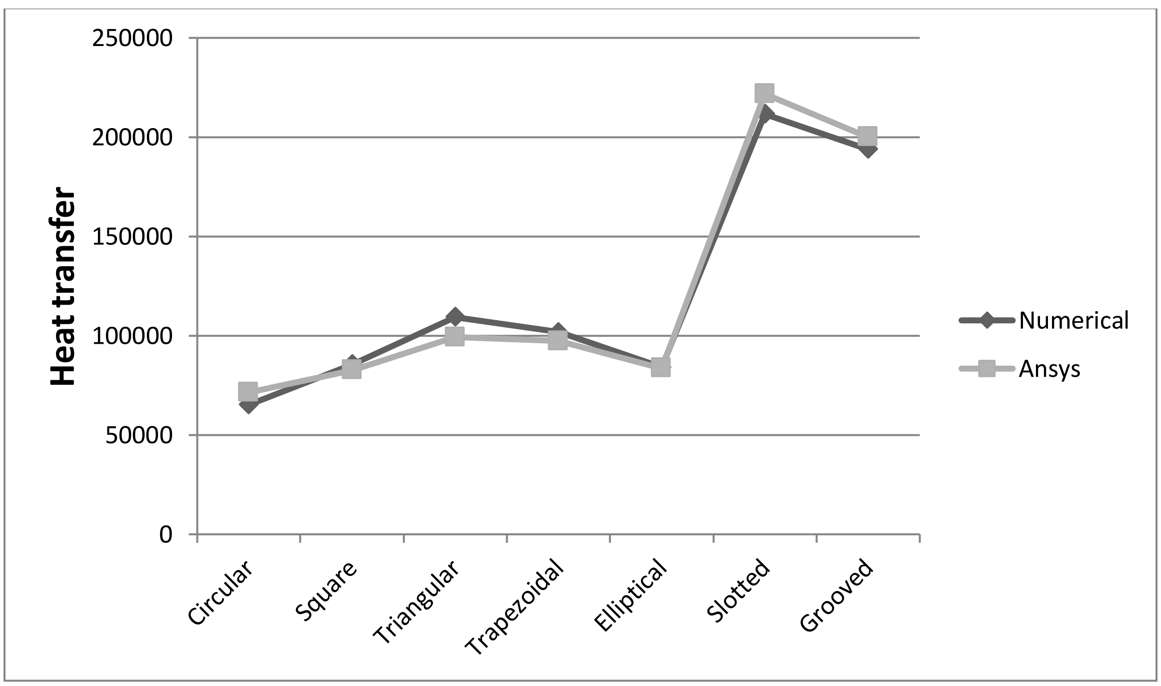


Fig.8.1. Comparison of heat transfer rate calculated

From the fig.8.1, it is clear the heat transfer is dependent on the thickness of the fin and is indirectly proportional. Hence in our analysis, the thinnest fin (i.e.) the fin with triangular cross section with a slot is having highest heat transfer as shown in the fig.8.1. The other attribute related to the heat transfer is the surface area of the fin. The more the surface area, more is the heat transfer to take place by convection. Hence, slotted fin and the grooved fin having higher surface areas compared to other fins have higher heat transfer rates as shown in fig.8.2. The heat transfer rates of slotted and grooved models increased by 93.3% and 77.2% compared to the triangular profile. The lower temperature at the end of the fin as shown in fig.8.3 is indicative of higher heat lost along the length of the fin. Hence better cooling is achieved for the engine. From the fig.8.3, the optimized fins have 2.8% and 1.47% lower temperature at tip. Thus the optimized fins can perform better to remove heat from the engine.

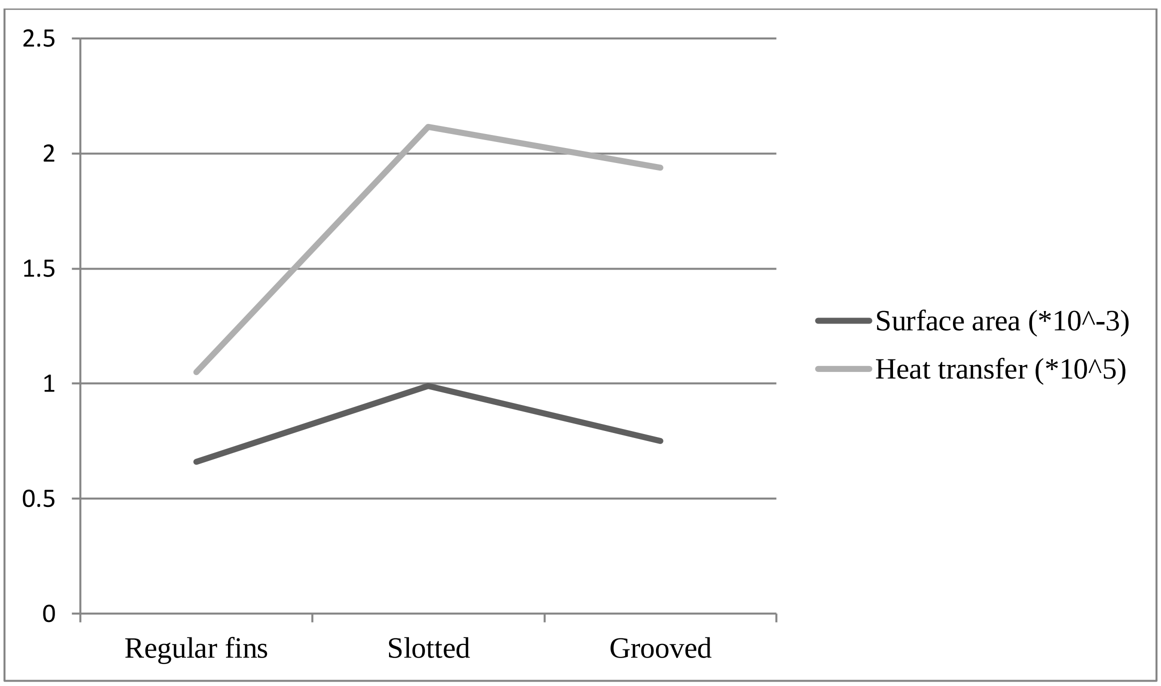


Fig.8.2. Heat transfer rate Vs Surface area of convection

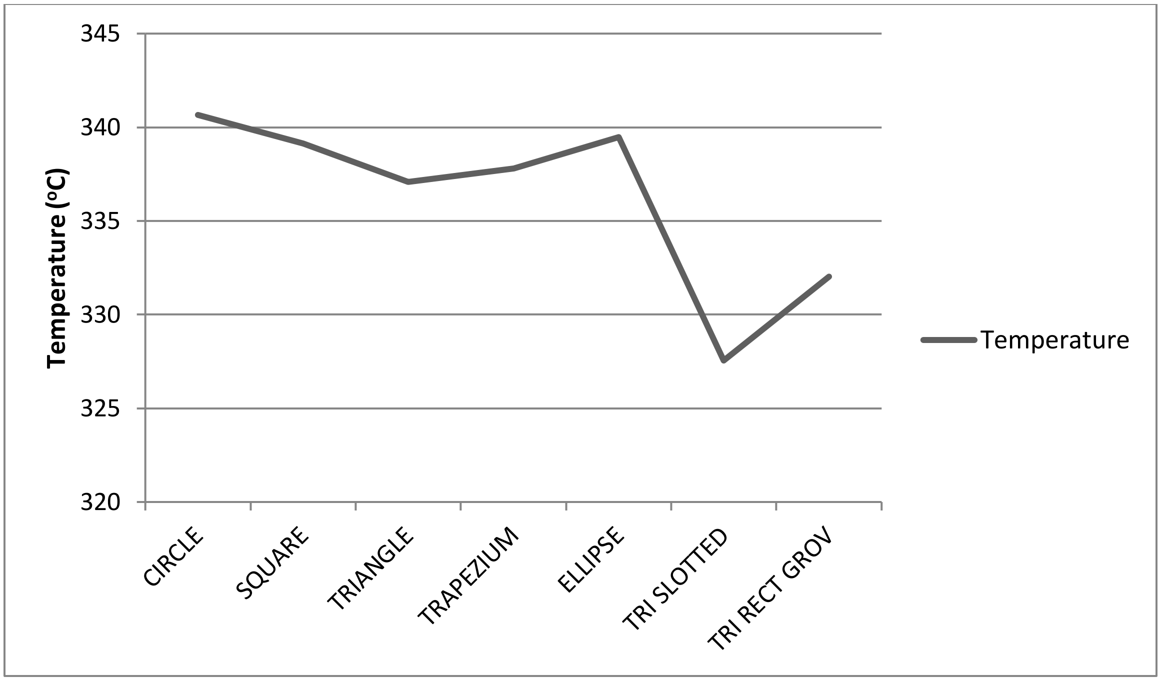


Fig.8.3. Temperature at X=30 mm

Additionally, it can be noted that the optimized design involves removal of material. This can in turn result in the reduction of mass of the cylinder. Thus, the optimized design also has the ulterior benefit of cost saving through material reduction. It is seen that there is 4.56% and 3.32% material reduction in the slotted and grooved models compared to the triangular cross section. The heat transfer rate vs. mass is shown in fig.8.4. It is clear that the slotted model having the highest heat transfer and least mass, can function better and material reduction wise economic. The next better option is grooved fins.

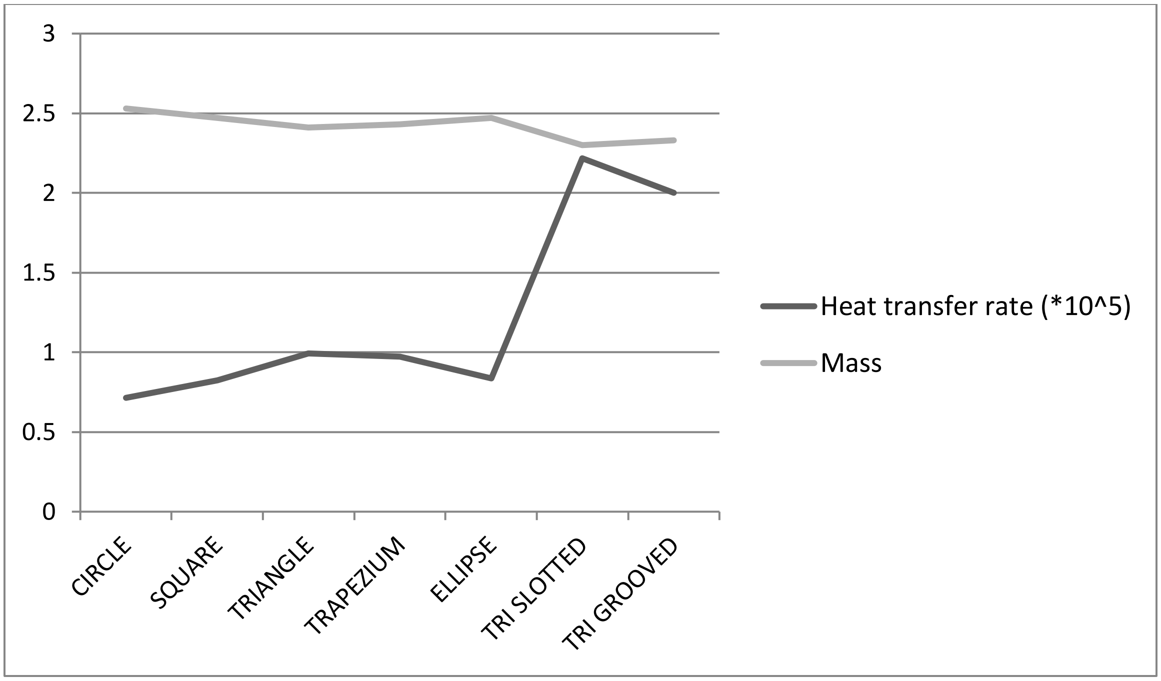


Fig.8.4. Heat transfer rate Vs Mass of cylinder

Future scopes for analysis:

The optimized models can be used in other places where the fins are employed such as transformers, heat sinks in electronic equipments, CPUs, etc….. and analyzed for performance. Additionally, the forced convection with air flow can be used for analysis. Moreover, the fins can be further optimized by utilizing other designs, changing materials, etc….. The fins can also be analyzed by the taking into consideration, the fins of varying area cross section unlike the constant area cross section discussed here.

SUMMARY

1. In this project, it is discussed what fins are and its applications in various fields.
2. An overview heat transfer methods involved and its boundary conditions.
3. Inspection of heat transfer from constant area fins of a modeled IC engine
4. Optimizing the heat transfer rate by increasing the surface area

5. Numerical analysis of heat transfer from fins
6. Finite elemental analysis using ANSYS.

REFERENCES:

1. Abdullah H. AlEssa (2009) 'Enhancement of natural convection heat transfer from a fin by rectangular perforations with aspect ratio of two'- International Journal of Physical Sciences Vol. 4 (10), pp. 540-547

2. Ajay Paul.J et.al. (2013) 'Experimental and Parametric Study of Extended Fins In The Optimization of Internal Combustion Engine Cooling Using CFD' - IOSR Journal of Mechanical and Civil Engineering Volume 7, Issue 4 (Jul. - Aug. 2013) ISSN: 2278-1684

3. Babu.G, M. Lavakumar (2013) 'Heat Transfer Analysis and Optimization of Engine Cylinder Fins of Varying Geometry and Material' - IOSR Journal of Mechanical and Civil Engineering e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 7, Issue 4, PP 24-29

4. C.P.Kothandaraman and S.Subramanyan (2016) 'Heat and Mass Transfer Data Book', New Age International Publishers

5. Dr. Kirpal Singh (2004) Automobile engineering Vol.II, Standard Publishers Distributors, Delhi

6. Mishra A.K., Nawal S. and Thundil Karuppa Raj R (2012) 'Heat Transfer Augmentation of Air Cooled Internal Combustion Engine Using Fins through Numerical Techniques' Research Journal of Engineering Sciences ISSN 2278 – 9472 Vol. 1(2), 32-40

7. Mohsin A. Ali (2014) 'DESIGN MODIFICATION AND ANALYSIS OF TWO WHEELER COOLING FINS-A REVIEW' - International Journal of Advances in Engineering & Technology, July, 2014 ISSN: 22311963

8. Phani Raja Rao.N et.al. (2013)’ Thermal Analysis Of Engine Cylinder Fins By Varying Its Geometry And Material’ - International Journal of Engineering Research & Technology ISSN: 2278-0181 Vol. 2 Issue 8

9. Prof. R.K. Rajput, “Heat and Mass Transfer”, S. Chand Publications.

10. Rameshkumar.A, Nandha.S Kumar, 2016, “Heat Transfer Analysis of Engine Cylinder Fin by Varying Extension Geometry” , Conference on contemporary approaches in Design and Manufacturing

11. Sai Chaitanya.P (2014)’Thermal Analysis of Engine Cylinder Fin by Varying Its Geometry and Material’ - IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 6 Ver. I (Nov- Dec. 2014), PP 37-44