STATIC STRESS, CONTACT STRESS AND THERMAL ANALYSIS ON A SPUR GEAR

A Project Report submitted in partial fulfilment of the requirements

for the award of the Degree of

BACHELOR OF TECHNOLOGY

in

MECHANICAL ENGINEERING

Submitted by

R.SAI PAVAN KALYAN	(315126520189)
M.NITHIN SAI MANIKANTA	(315126520156)
N.YESWANTH REDDY	(315126520149)
P.J.J.SRIKAR	(315126520183)
P.VENKATESH	(315126520174)

Under the guidance of

Dr.K.SATYANARAYANA

M.Tech,Ph.D

Assistant professor, Mechanical Engineering Department

DEPARTMENT OF MECHANICAL ENGINEERING

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY &SCIENCES

(UGC-AUTONOMOUS)

(Affiliated to A.U., approved by AICTE, Accredited by NBA and NAAC
with 'A'grade)

SANGIVALASA, VISAKHAPATNAM-531162

(2015 - 2019)

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY &SCIENCES

(UGC-AUTONOMOUS)

(Affiliated to A.U., approved by AICTE, Accredited by NBA and NAAC
with 'A'grade)

SANGIVALASA, VISAKHAPATNAM-531162

DEPARTMENT OF MECHANICAL ENGINEERING

CERTIFICATE

This is to certify that the project entitled "**STATIC STRESS, CONTACT STRESS AND THERMAL ANALYSIS ON A COMPOSITE SPUR GEAR**" being submitted by **R.SAI PAVAN KALYAN(315126520189), M.NITHIN SAI MANIKANTA(315126520156), N.YESWANTH REDDY(315126520149), P.J.J.SRIKAR (315126520183), P. VENKATESH (315126520174)** in partial fulfillment of the requirement for the award of Degree of "**BACHELOR OF TECHNOLOGY**" in **MECHANICAL ENGINEERING.** It is bona fide work carried out under the guidance and supervision of Dr. K.SATYANARAYANA, Assistant Professor of Department of Mechanical Engineering, during the academic year of 2018-2019.

APPROVED BY

(Prof. B. Naga Raju)

Head of the Department,

Dept. of Mechanical Engineering

ANITS, Sangivalasa

Visakhapatnam.

PROJECT GUIDE

15/4/19.

Dr. K .Satyanarayana

Asst. Professor

Dept. of Mechanical Engineering

ANITS, Sangivalasa

Visakhapatnam.

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE'
Sangivalasa 531 162 VISAKHAPATNAM Dist A P

THIS PROJECT WORK IS APPROVED BY THE FOLLOWING BOARD OF EXAMINERS

INTERNAL EXAMINER:

15. 4. 19

EXTERNAL EXAMINER:

15/04/19

ACKNOWLEDGEMENT

ABSTRACT

Spur gears are the simplest type of gears, which are widely used in modern structures. One of the main task of the researchers is to obtain compact gear, small size and weight, while maintaining or improving their functional characteristics. Therefor there was a need for exploration of new materials that could be used as a replacement for structural steel in the manufacturing of gears.

In this project we designed a spur gear of standard dimensions on design software and Static structural and thermal analysis is done for the structural steel material, Grey cast iron, Nylon, CFRP, AlSiC15%, AlSiC16% CNT1% composite materials and the results are compared with each other and the best material is selected. Also the contact stress analysis is done for the structural steel. In this particular project Modelling and Assembly are done using **CATIA V5** software and the analysis is carried out using **ANSYS workbench19.2** for determining the Equivalent stresses, Deformations, Temperature distribution, Contact stresses and Heat flux distribution **.**

CHAPTER 1

Introduction

1.1 DEFINITION OF GEARS

Gears are toothed wheel members which transmit power or motion between two shafts by meshing without any slip. Hence, gear drives are also called as positive drives. In pair of gears, the smaller one is called pinion and the larger one is called as gear immaterial of which is driving the other. When pinion is the driver, it results in step down drive in which the output speed decreases and the torque increases. On the other hand, when the gear is the driver, it results in step up drive in which the output speed increases and the torque decreases.

1.2 CLASSIFICATION OF GEARS

Classification of gears can be done according to relative position of the axes of revolution into three types. They are

- **Gears for parallel shafts**
  - Spur gears
  - Helical gears
  - Herringbone gears
- **Gears for intersecting shafts**
  - Straight bevel gears
  - Spiral bevel gears
- **Gears for skew shafts**
  - Hypoid gears
  - Worm gears

1.2.1 Gears for parallel shafts

The motion between parallel shafts is same as to the rolling of two cylinders. Gears under this category are the following:

1.2.1.1 Spur gears

Straight spur gears are the simplest form of gears having teeth parallel to the gear axis. The contact of two teeth takes place over the entire width along a line parallel to the axes of rotation. As gear rotates, the line of contact goes on shifting parallel to the shaft.

Advantages of spur gears:

o   No axial thrust is created by the tooth loads.
o   Spur gears are excellent at moderate speeds.
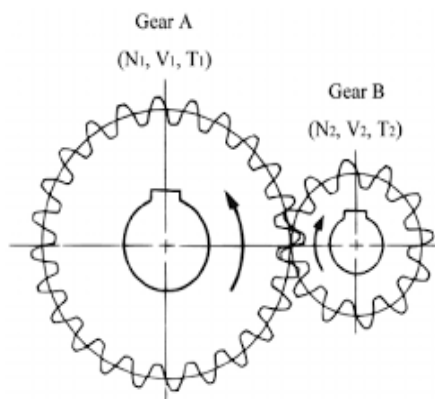
Figure 1.1 External Spur Gear   Figure 1.2 Internal Spur Gear

1.2.1.2 Helical gears

In helical gears the teeth are part of helix instead of straight across the gear parallel to the axis. The mating gears will have same helix angle but in opposite direction.
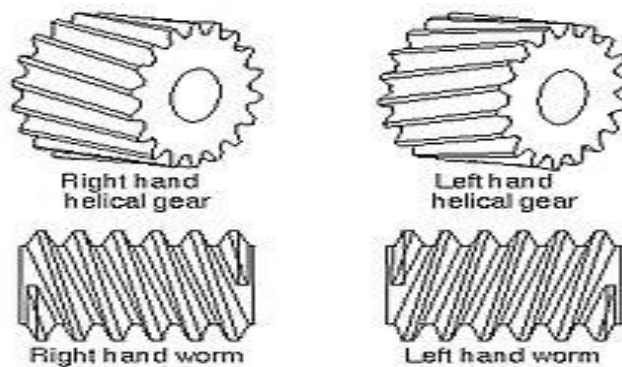
Figure 1.3 Helical Gear

1.2.1.3 Herringbone gear

Herringbone gears are also known as **Double Helical Gears.** Herringbone gears are made of two helical gears with opposite helix angles, which can be up to 45 degrees.

1.2.1.4 Rack and Pinion

In these gears the spur rack can be considered to be spur gear of infinite pitch radius with its axis of rotation placed at infinity parallel to that of pinion. The pinion rotates while the rack translates.
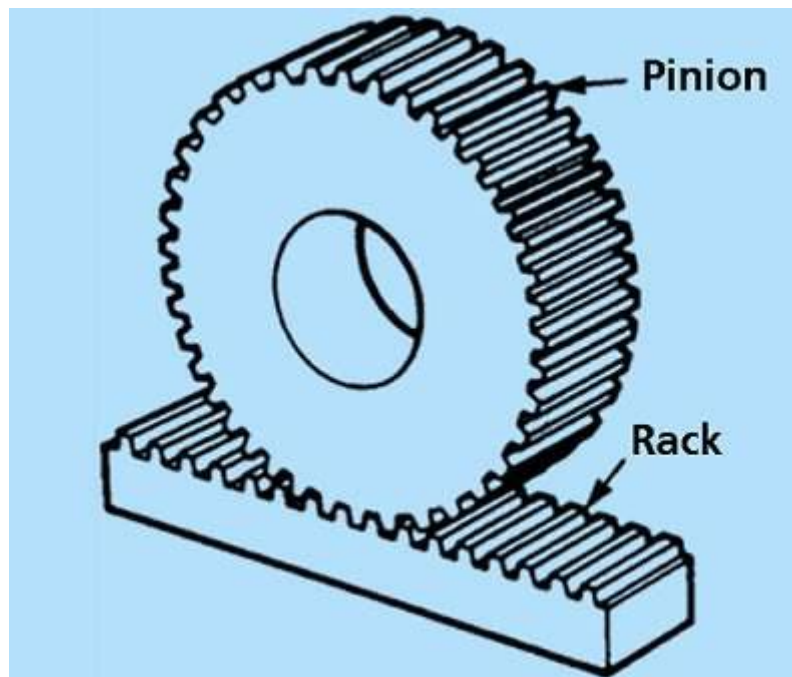
Figure 1.4 Rack and Pinion

1.2.2 Gears for intersecting shafts

The motion between two intersecting shafts is equivalent to the rolling of two cones. The gears used for intersecting shafts are called bevel gears. Gears under this category are as follows:

1.2.2.1 Straight Bevel Gears

Straight bevel gears are provided with straight teeth, radial to the point of intersection of the shaft axes and vary in cross section through the length inside the generator of the cone. Straight Bevel Gears can be seen as modified version of straight spur gears in which teeth are made in conical direction instead of parallel to axis.

1.2.2.2 Spiral Bevel Gears

Bevel gears are made with their teeth are inclined at an angle to face of the bevel. Spiral gears are also known as helical bevels.
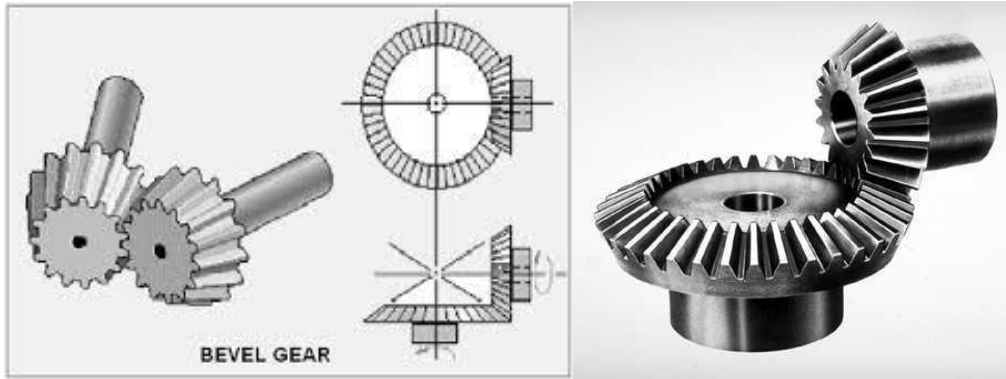

Figure 1.5 Bevel Gear

1.2.3 Gears for Skew Shafts

The following gears are used to join two non-parallel and non-intersecting shafts.

1.2.3.1 Hypoid Gears

The Hypoid Gears are made of the frustum of hyperboloid of revolution. Two matching hypoid gears are made by revolving the same line of contact, these gears are not interchangeable.

Figure 1.6 Hypoid Gear

1.2.3.2 Worm Gears

The Worm Gears are used to connect skewed shafts, but not necessarily at right angles. Teeth on worm gear are cut continuously like the threads on a screw. The gear

meshing with the worm gear is known as worm wheel and combination is known as worm and worm wheel.
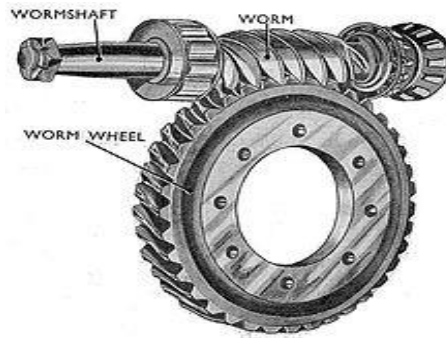

Figure 1.7 Worm Gear

1.3 BASIC GEAR TERMINOLOGY

The following are the important dimensions and geometries concerned with toothed gear.
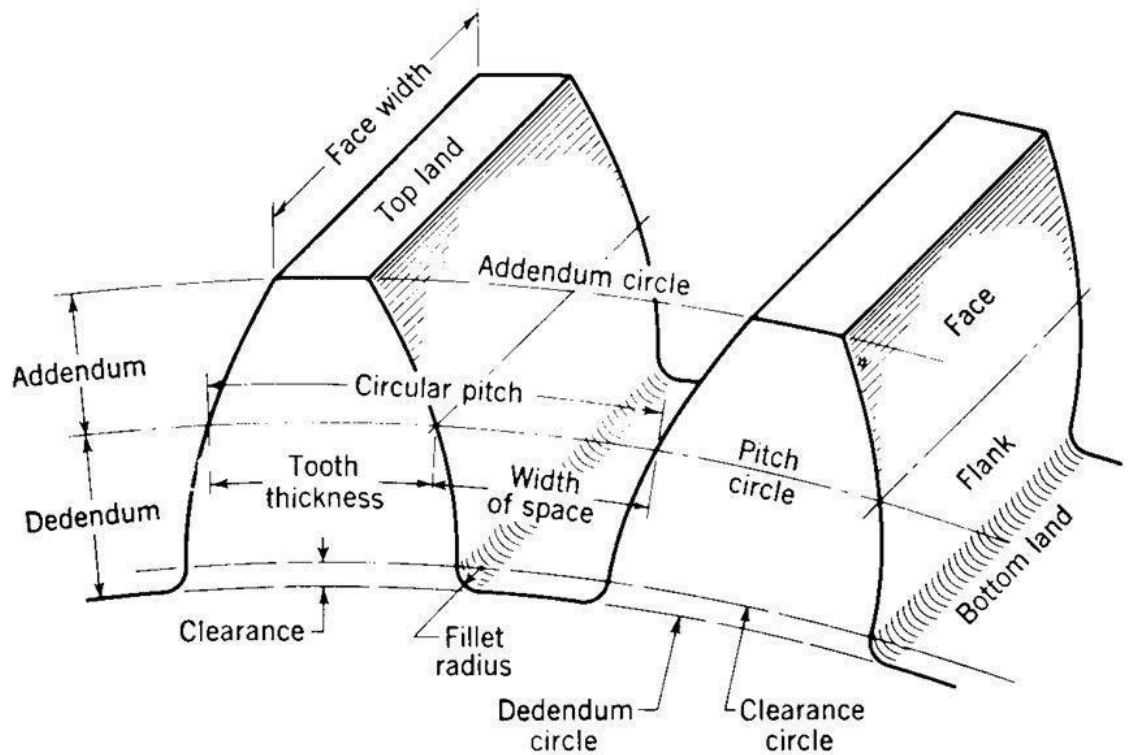

Figure 1.8 Gear Nomenclature

1.3.1 Pitch Circle

Pitch circle is the apparent circle that two gears can be taken like smooth cylinders rolling without friction.

1.3.2 Addendum Circle

Addendum circle is the outer most profile circle of a gear. Addendum is the radial distance between the pitch circle and the addendum circle.

1.3.3 Dedendum Circle

Dedendum circle is the inner most profile circle. Dedendum is the radial distance between the pitch circle and the dedendum circle.

1.3.4 Clearance

Clearance is the radial distance from top of the tooth to the bottom of the tooth space in the mating gear.

1.3.5 Backlash

It is the difference between the tooth space and the tooth thickness, as measured on the pitch circle.

1.3.6 Full Depth

Full Depth is the sum of addendum and the dedendum.

1.3.7 Face Width

Face width is the length of tooth parallel to axes.

1.3.8 Diametrical Pitch

It is the ratio of number of teeth to the pitch circle diameter in millimetres. It is denoted by P_d.

1.3.9 Module

It is the ratio of the pitch circle diameter iin millimetres to the number of teeth. It is usually denoted by **m.**

$$m=d/z$$

where d=pitch circle diameter

z=number of teeth

1.3.10 Circular Pitch

It is the distance measured on the circumference of the pitch circle from a point of one tooth to the corresponding point on the next tooth. It is usually denoted by P_c.

1.3.11 Gear Ratio

Gear ratio is the number of teeth of larger gear to the number of teeth of smaller gear.

1.3.12 Pressure Line

Pressure line is the common normal at the point of contact of mating gears along which the driving tooth exerts force on the driven tooth.

1.3.13 Pressure angle

Pressure angle is the angle between the common normal drawn at point of contact and common tangent of pitch circles at pitch point. It is also called as **Angle of Obliquity.** High pressure angles requires wider base and stronger teeth.

1.3.14 Pitch angle

It is the angle made by the pitch line with the axis of the shaft. It is denoted by $\boldsymbol{\theta p}$.
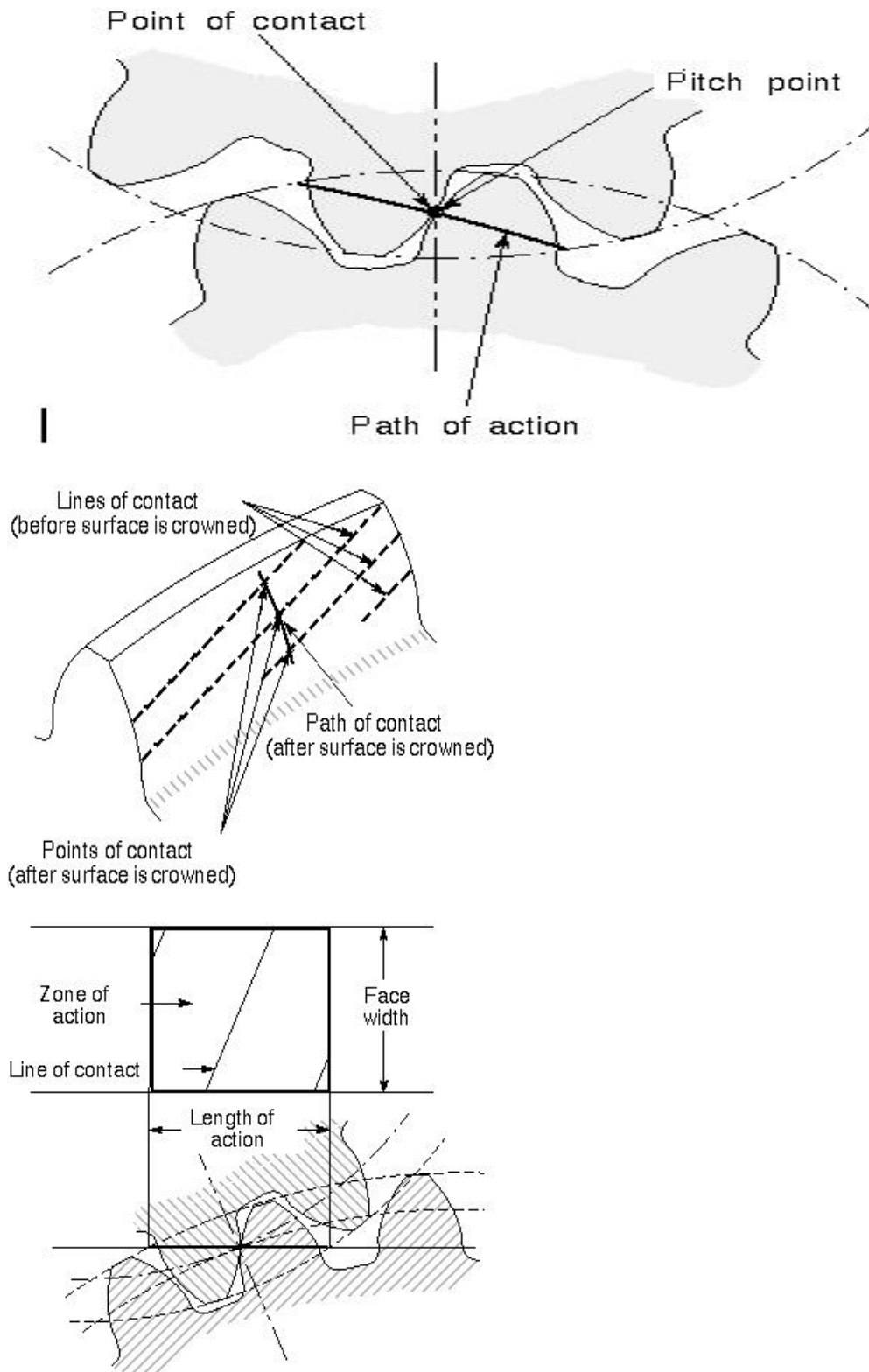
1.4 TOOTH CONTACT NOMENCLATURE

Figure 1.9 Gear Tooth Contact Nomenclature

1.4.1 Point of contact

Any point at which two tooth profiles touch each other

1.4.2 Line of contact

A line or curve along which two tooth surfaces are tangent to each other.

1.4.3 Path of action

The locus of successive contact points between a pair of gear teeth, during the phase of engagement. For conjugate gear teeth, the path of action passes through the pitch point. It is the trace of the surface of action in the plane of rotation.

1.4.4 Line of action

The path of action for involute gears. It is the straight line passing through the pitch point and tangent to both base circles.

1.4.5 Surface of action

The imaginary surface in which contact occurs between two engaging tooth surfaces. It is the summation of the paths of action in all sections of the engaging teeth.

1.4.6 Plane of action

The surface of action for involute, parallel axis gears with either spur or helical teeth. It is tangent to the base cylinders.

1.4.7 Zone of action (contact zone)

For involute, parallel-axis gears with either spur or helical teeth, is the rectangular area in the plane of action bounded by the length of action and the effective face width.

1.4.8 Path of contact

The curve on either tooth surface along which theoretical single point contact occurs during the engagement of gears with crowned tooth surfaces or gears that normally engage with only single point contact.

1.4.9 Length of action

The distance on the line of action through which the point of contact moves during the action of the tooth profile.

1.4.10 Arc of action

The arc of the pitch circle through which a tooth profile moves from the beginning to the end of contact with a mating profile. It is denoted by Q_t

1.4.11 Arc of approach

The arc of the pitch circle through which a tooth profile moves from its beginning of contact until the point of contact arrives at the pitch point. It is denoted by Q_a

1.4.12 The arc of a recess

The arc of the pitch circle through which a tooth profile moves from contact at the pitch point until contact ends. It is denoted by Q_r.

1.5 LAW OF GEARING

Law of gearing states that common normal to the tooth profile at the point of contact should always pass through a fixed point called pitch point in order to obtain a constant velocity ratio.
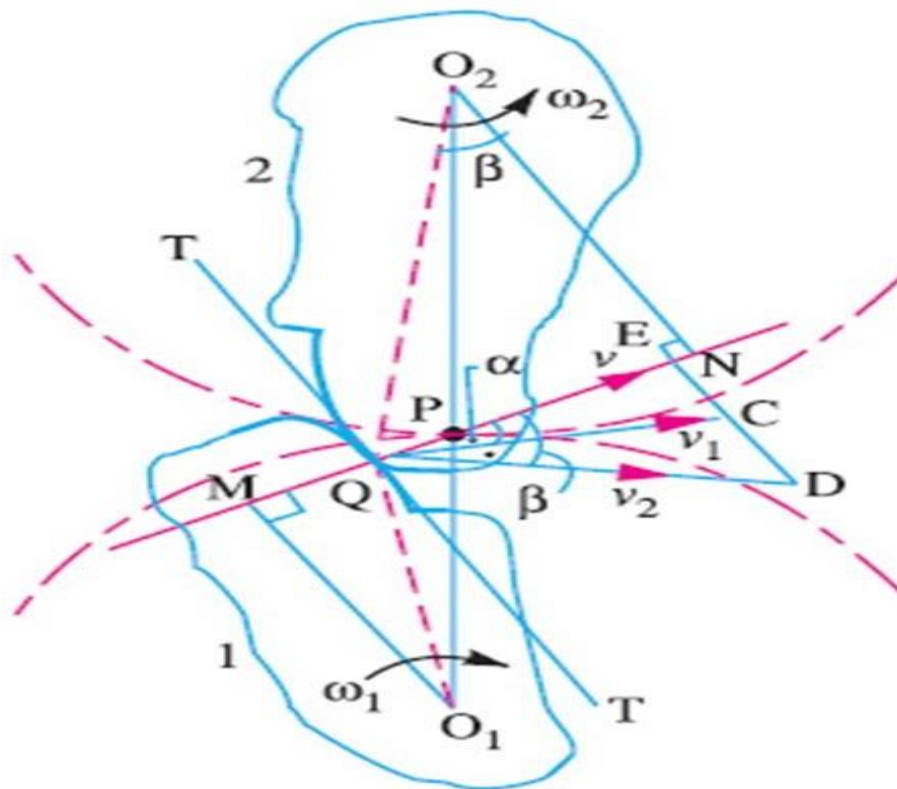
Figure 1.10 Law Of Gearing

where gear wheel-1 is considered point Q moves in the direction of QC. Velocity along QC is V_1. Where gear wheel-2 is considered point Q moves in the direction QD.

Velocity along QD is V_2. As the teeth are to remain in contact, then the components of these velocities along the common normal MN must be equal.

1.5.1 Derivation Of Law Of Gearing

$$V = v_1\cos\alpha = v_2\cos\beta \ldots\ldots..1$$

$$V_1 = O_1Qw_1 \ldots\ldots..2$$

$$V_2 = O_2Qw_2\ldots\ldots.3$$

Substituting 2 and 3 in 1 we get,

$$V = O_1Qw_1\cos\alpha = O_2Qw_2 \cos\beta \ldots\ldots..4$$

$$\cos\alpha = O_1M/ O_1Q \ldots\ldots\ldots5$$

$$\cos\beta = O_2N/O_2Q \ldots\ldots\ldots6$$

Substituting 5 and 6 in 4 we get,

$$O_1Qw_1\times O_1M/ O_1Q = O_2Qw_2\times O_2N/O_2Q$$

$$w_1/ w_2 = O_2N/ O_1M \ldots\ldots\ldots.7$$

Consider similar triangles O_1MP and O_2NP as shown below,

$$O_2N/ O_1M = d_2/d_1 = Z_2/Z_1\ldots\ldots\ldots8$$

Substituting 8 in 7 we get,

$$w_1/ w_2 = Z_2/Z_1$$

Hence it is proved that velocity ratio is constant.
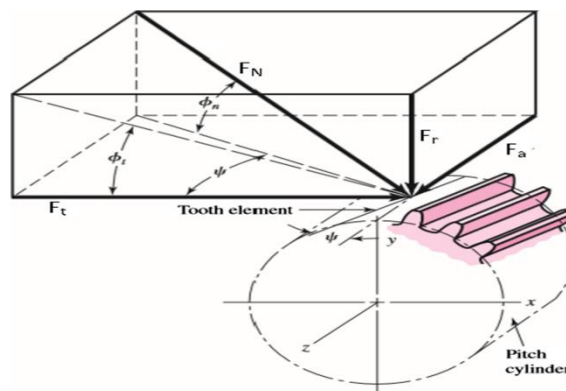
1.6 LEWIS EQUATION FOR BENDING STRESS

Figure 1.11 Representation of Forces on Gear Tooth

Lewis considered gear tooth as a cantilever beam with static normal force F applied at the tip.

Assumptions made in the derivation are:

1. The full load is applied to the tip of a single tooth in static condition.

2. The radial component is negligible.

3. The load is distributed uniformly across the full face width.

4. Forces due to tooth sliding friction are negligible.

5. Stress concentration in the tooth fillet is negligible.

At point 'a', bending stress is

$$\sigma = Mc/I = 6*F_t*h/(b*t^2) \ldots\ldots..1$$

by similar triangles,

$$t^2/h = 4*X \ldots\ldots.2$$

Substituting of Eqn. 2 in Eqn. 1, it gives

$$\sigma = 6*F_t/(4*b*x) \ldots..3$$

$$y = 2*x/(3*p) \ldots..4$$

where y is defined as the lewis form factor

and substituting equation 4 in 3 we get,

$$\sigma = F_t/(p*b*y) \ldots\ldots.5$$

Eqn. 5 is the basic lewis equation in terms of circular pitch

But P=π*m

$$\sigma = F_t/(\pi*m*b*y) \ldots\ldots.6$$

let Y = y* π, known as modified lewis form factor

$$\sigma = F_t/(b*Y*m) \ldots..7$$

Eqn.7 is the standard equation for tooth bending stress based on module.

1.7 MODES OF FAILURE IN GEARS

Gear Systems or gear trains tend to play a very vital role in all industries and also in our day to day life. Any failure to the gear system leads to the total system failure. Imagine driving a car with a broken gear in its gearbox. Knowing the various modes by which the gears tend to fail helps one to identify the problem at an initial stage and rectify the problem related to the gear. Now let us look into the various modes by which the gears fail:

The various major areas by which the gears tend to fail are

- Shock Loading Failures
- Fatigue Failures
- Failures due to Wear
- Failures due to Scuffing

1.7.1 SHOCK LOADING

- **Breakage by sudden shock loads**: When the gear system is subjected to a sudden shock load more than it is designed for, it instantly breaks. Though this cannot be identified prematurely, utmost care can be taken to avoid such mishaps.

1.7.2 FATIGUE FAILURES

- **Bending Fatigue**: Gears based on their rotational motion and the heavy amount of load they carry are subjected to fatigue. Due to the application of load on the gear teeth, the gear tooth is subjected to bending. When the same is repeated for a very huge number of cycles Bending Fatigue failure of the gear sets in. The fatigue in the gears induces the formation of cracks in the root of the gear tooth which propagates with each rotational cycle of the gear and ultimately leads to the failure of the gear tooth.

  1.7.3 Pitting: This is a major cause of gear failure accounting for nearly 60% of the gear failures. Pitting is the formation of craters on the gear tooth surface. These craters are formed due to the high amount of compressive contact stresses in the gear surface occurring during transmission of the torque or in

simple terms due to compressive fatigue on the gear tooth surface. There are two types of Pitting. They are

1.7.3.1 Micro Pitting:

These are basically formed due to

- o Inherent Errors in the gears
- o Presence of water in the lubricant that is lubricating the gears
- o Wrong viscosity selection of the lubricant used.
- o Visually, micro pitting is not so clearly visible at the first go. One has to study the surface of the gear tooth to identify the micro-pitting. They appear as very small dots which one can feel when he runs his finger over the gear tooth.

- This sort of pitting normally tends to make the gear useless and damages the whole gear system.

1.8 Gear materials

1.8.1 Structural Steel

Structural steel is a category of steel construction material that is produced with a particular cross section or shape and some specified values of strength and chemical composition. The word structural includes a broad variety of low carbon and manganese steels that are used for numerous applications in civil and marine engineering applications.

1.8.1.1 Types of Structural Steel

After iron carbon is the most important element in the steel. Increase in carbon content increases the strength of the material and decreases its ductility.

1. Carbon Manganese Steels:

    The major chemical ingredients are iron, carbon and manganese. These are also called as **Carbon steels**. These have high strength and high ductility.

2. High strength, Low-alloy steel:

    This is a recent development in industry. Chemical elements area added to improve strength.

3. High strength tempered and quenched alloy steel:

These are especially used for structural purposes.

1.8.1.2 Properties of Structural steel

Young's module: 207Gpa

Poisson's Ratio: 0.3

Density:7.85 g/cm^3

Yield strength: 207Mpa

Tensile strength: 345Mpa

1.8.2 Grey Cast Iron

It is a cast iron structure that have graphite in it. It is named after the grey color after fracture which is caused due to the presence of graphite in it. It is used where the stiffness of the component is more required. It is the common Engineering alloy because of its low cost and good machinability. It also has good wear resistance. It also experiences less solidification shrinkage.

1.8.2.1 Properties of Grey Cast iron

Young's module: 13.4Gpa

Poisson's Ratio: 0.3

Density:7.15 g/cm^3

Yield strength: 90Mpa

Tensile strength: 140Mpa

1.8.3 Composite materials

A composite material can be defined as a combination of two or more materials that results in better properties than those of the individual components used alone. The main advantages of composite materials are their high strength and stiffness, combined with low density, when compared with bulk materials, allowing for a weight reduction in the finished part.

1.8.3.1 Carbon Fiber Reinforced composites

Carbon fiber reinforced composite is an extremely strong metal with less weight. These are expensive to produce but are used when there is a requirement of high strength to weight ratio. The binding polymer is generally a thermoset resin such as epoxy but other thermoset resins such as Nylon, Polyster are generally used.in this case the material contains two parts: a matrix and a reinforcement. In CFRP reinforcement is carbon fiber which gives strength and the matrix is usually a polymer which increases the binding ability.

1.8.3.1.1 Properties of CFRP

Young's module: 132Gpa

Poisson's ratio:0.39

Density:1.43 g/cm^3

Yield strength:300Mpa

Tensile strength: 577Mpa

1.8.3.2 Nylon

Nylon is a generic designation for a family of synthetic polymers. Nylon is a thermoplastic silky material that can be melt processed into fibers. Nylon can be added with a different varieties of additives to get various properties.

1.8.3.2.1 Properties of Nylon

Young's module: 225Gpa

Poisson's ratio: 0.3

Density: 1.13 g/cm^3

Yield strength: 82Mpa

Tensile strength: 82Mpa

1.8.3.3 Aluminum – Silicon carbide composite

Aluminum - silicon carbide is a metal ceramic composite material consisting of silicon carbide in a matrix of aluminum alloy. It combines the benefits of high thermal conductivity of metal and low thermal coefficient of expansion of ceramic.

1.8.3.3.1 Properties of Al, SiC 15%

Young's module: 150Gpa

Poisson's ratio: 0.3

Density: 2.79 g/cm^3

Yield strength: 90Mpa

Tensile strength: 151Mpa

1.8.3.3.2 Properties of Al, SiC 16%, CNT 1%

Young's module: 93Gpa

Poisson's ratio: 0.3

Density: 2.79 g/cm^3

Yield strength: 150Mpa

Tensile strength: 202Mpa

CHAPTER 2

LITERATURE REVIEW

The background research work for the project is done to obtain details from various other journals regarding the design specifications and material usage. At first search is done for the type of gear to be selected, then after selecting the type of gear our focus shifted to the type of alternative materials that can be used and at last the theoretical calculations for various stresses.

2.1 Literature Review

Slobodan Garicl [1]: Spur gears are the simplest type of gears, which are widely used in modern structures. One of the main tasks of researchers in application areas of gears within the responsible structures, especially in spacecraft propulsion mechanisms, is to obtain compact gear, small size and weight, while maintaining or improving functional characteristics. Therefore, there was a need to explore new materials that could be used as a replacement for steel in the manufacturing of gears. A good alternative for steel is hybrid composite with aluminum matrix and various reinforcements, which provides adequate mechanical properties while reducing the weight of the gear. The paper presents the modeling of gears and analysis of their stress-deformation state using the finite element method. Steel and composite gears made from the aluminum matrix with various reinforcements were tested. Results from this study show that the gear made of composite material offers enhanced features compared to metal gears

Mr. Kailas S. Pathade1, Prof. Santosh Mali Patil [2]: This paper presents the work carried regarding the determination of true bending stresses in spur gears. Gearing is one of the most critical components in a mechanical power transmission system and the true bending stresses at the tooth root of spur gears are quite different from the nominal values. The three-dimensional models of a set of spur gear were prepared using 'KISSsoft' software. Simulation is done in ANSYS workbench and the results are compared.

Ismail Ali Abdul Aziz1 Daing Mohamad Nafiz Daing Idris1, and Wafiuddin MohdGhazali1 [3]: This article studies the methodology used to investigate bending strength of spur gear; Finite Element Method (FEM), Numerical Calculation and Investigational Techniques were usually carried out in order to understand the bending

strength of thin-rimmed spur gear. The most common method used to investigate bending strength is the numerical calculation.

Tushar Jain [4]: This paper deals with the bending stress calculation of spur gear theoretically and validating it with ANSYS software. These meshed gears from the gear box are subjected to considerable damage throughout its life span due to dynamic excitations caused by terrain undulations, the rotating wheel and track assemblies. The friction between two rolling gears results in the generation of heat. Due to this heat generation gears goes into plastic state and gets deformed well before its ultimate tensile strength. The sequence followed is first drafting of spur gear on SOLIDWORKS Next step is Static structural analysis which determines the bending stress generated in spur gear using FEA software ANSYS. The present work deals with the calculation of static analysis, and thermal analysis of gear by using ANSYS software. In a nutshell, this paper explicitly deals with spur gear meshing theoretical calculation after drafting it and validating it by using ANSYS software. The practical implication of this paper is significant as the gear train is a major driver in power transmission systems.

Shiferaw Damtie and Daniel Tilahun [5]: In this paper, the contact stress in rolling-sliding contact of the involute spur gear and the effect of the coefficient of friction was analyzed. Fem and Hertzian stresses are used to obtain the stresses. Modelling and simulation are done using SOLID WORKS and ANSYS respectively.

S.Mahendran [6]: To design the spur gear to study the weight reduction and stress distribution for cast steel and composite materials. Gearing is one of the most critical components in a mechanical power transmission system, and in most industrial rotating machinery. It is possible that gears will predominate as the most effective means of transmitting power in future machines due to their high degree of reliability and compactness. In addition, the rapid shift in the industry from heavy industries such as shipbuilding to industries such as automobile manufacture and office automation tools will necessitate a refined application of gear technology. Our aim is to design the spur gear model using designs software, study the impact analysis for cast steel and composite materials. To study the torque loading for cast steel and composite materials. Finally, comparing and analyzing of the composite gear with existing cast steel gear is to be done.

CHAPTER 3

THEORITICAL CALCULATIONS

3.1 SPECIFICATIONS OF THE GEAR AND PINION

In order to calculate the bending stresses on spur gears the specifications of spur gears and the properties of different materials are taken.

1. **Input strength, P** - **5 KW**
2. **Input speed, n_1** - **1440 rpm**
3. **Module, m** - **10 mm**
4. **Pressure angle, Ø** - **20°**
5. **Number of teeth on gear, N_1-** **40**
6. **Number of teeth on pinion, N_2.-** **20**
7. **Fillet** - **0.39 mm**

3.2 CALCULATION OF GEAR PARAMETERS

3.2.1 Pitch circle radius

$$R_p = m*N/2$$

Pitch circle radius of gear = R_{pg}=200 mm

Pitch circle radius of pinion= R_{pp}=100 mm

3.2.2 Base circle radius

$$R_b = 0.94*R_p$$

Base circle radius of gear = R_{bg}=188 mm

Base circle radius of pinion = R_{bp}=94 mm

3.2.3 Addendum circle radius

$$R_a = R_p + m$$

Addendum circle radius of gear = R_{ag}=210 mm

Addendum circle radius of pinion = R_{ap}=110 mm

3.2.4 Dedendum circle radius

$R_d = R_p - 1.25*m$

Dedendum circle radius of gear = $R_{dg} = 187.5$ mm

Dedendum circle radius of pinion = $R_{dp} = 87.5$ mm

3.2.5 Top angle

$T_a = 90/N$ deg

Top angle of gear = $T_{ag} = 2.25$ deg

Top angle of pinion = $T_{ap} = 4.5$ deg

3.2.6 Gear ratio, i

$i = N_1/N_2 = 2$

CHAPTER 4

CATIA INTRODUCTION

4.1 INTRODUCTION TO CATIA

CATIA, stands for **Computer Aided Three-dimensional Interactive Application**, it is the most powerful Knowledge based and widely used CAD (computer aided design) software of its kind in the world.

CATIA has been created by Dassault Systems of France and is marketed & technically supported worldwide by IBM. The most commonly CATIA users are generally Aerospace, Appliances, Architecture, Automotive, Construction, Consumer Goods, Electronics, Medical, Furniture, Machinery, Mold and Die, and Shipbuilding industries.

CATIA has played a major role in NASA's design of the various Space equipments. Beside this it has also been used as Vital tool for designing "jet-fighter" aircraft, aircraft carriers, helicopters, tanks and various other forms of weaponry extensively used by the Defense Sector.

CATIA is used throughout the North American and European continents, as well as Australia. Apart from this CATIA is increasingly being used by Asian countries like India, Japan etc.

Fig 4.1 Catia Interface
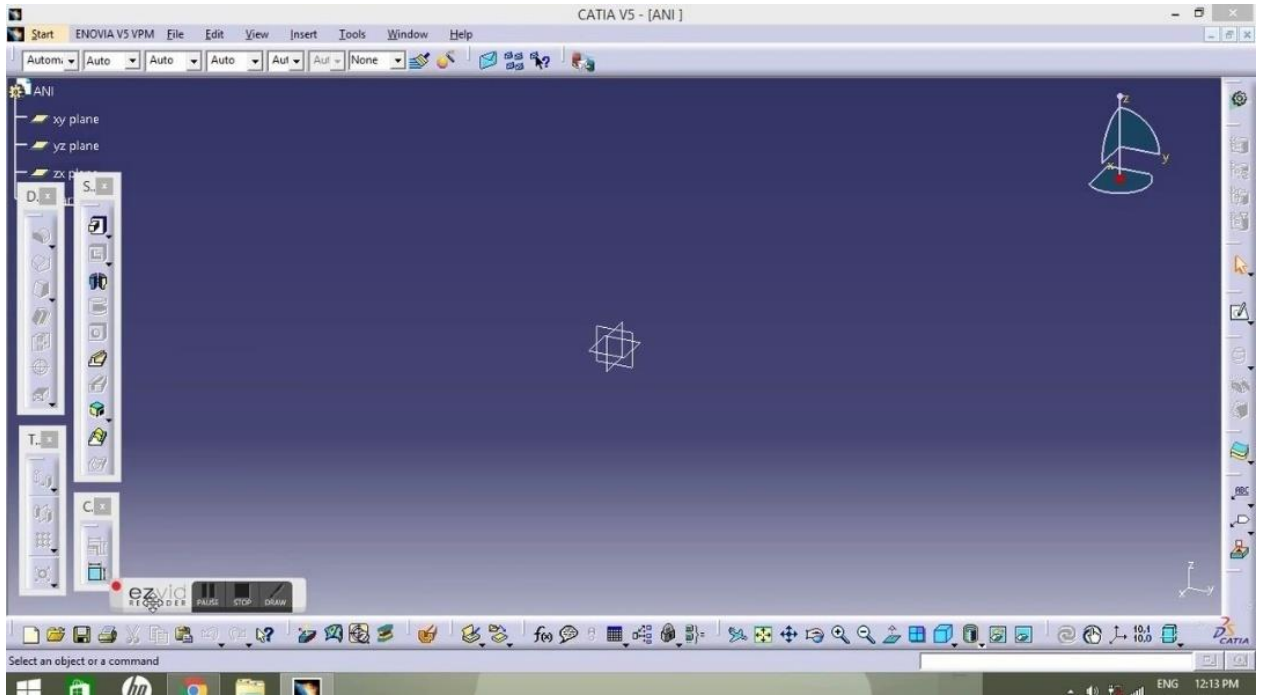
Fig 4.2 Catia Workbench

French Dassault Systems is the parent company and IBM participates in the softwares and marketing, and catia invades broad industrial sectors, and has been explained in the previous post position of CATIA between 3d modeling software programs.

Fig4.3 Different Modules in Catia

4.2 Sketcher:

This module is responsible for the implementation of two-dimensional shapes, in preparation for make a three-dimensional commands on it.
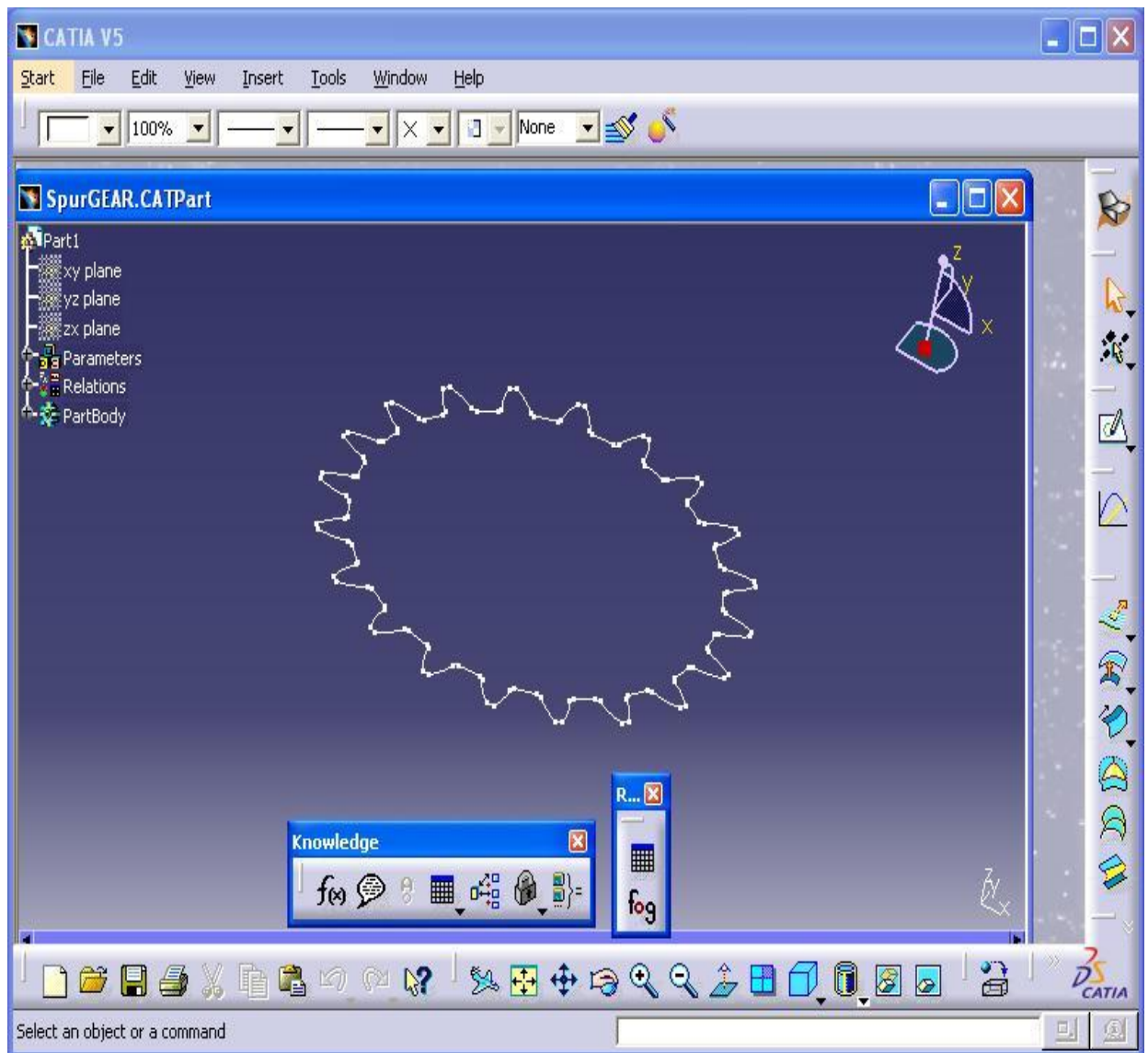
Fig4.4 Sketcher of gear in Catia

4.3 Part Design:

This module is 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 programs, which is stereotaxic drawing or three-dimensional drawing.
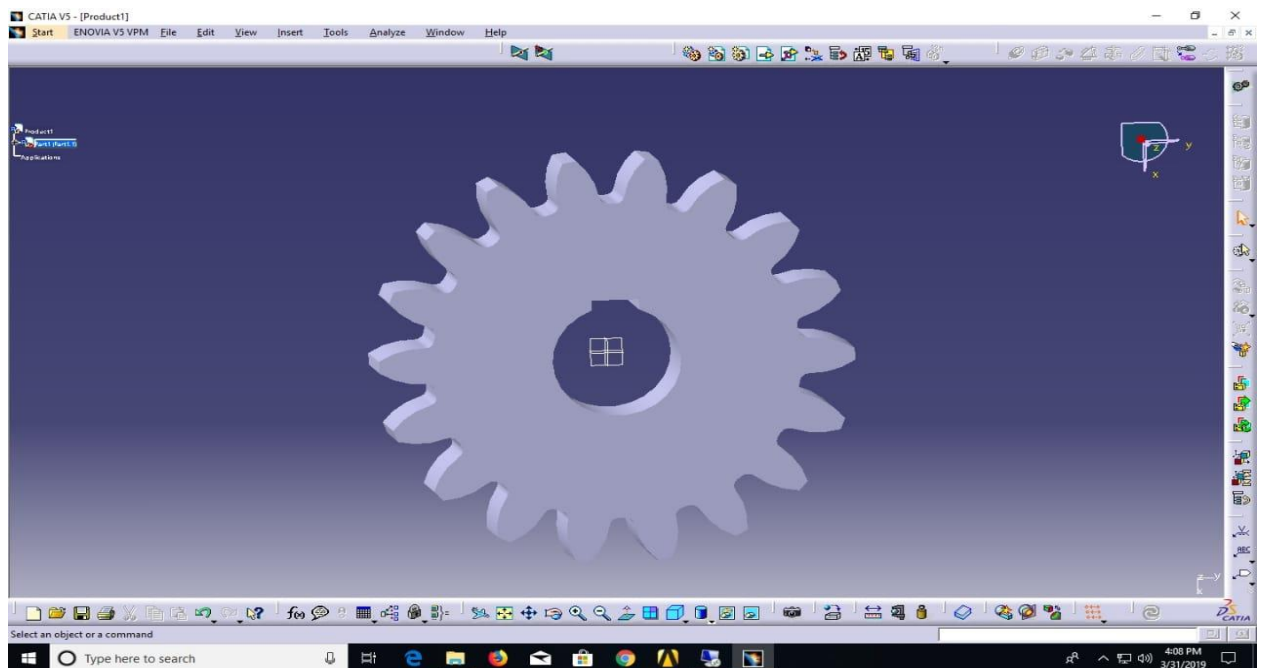
Fig4.5 Part Design of gear in Catia
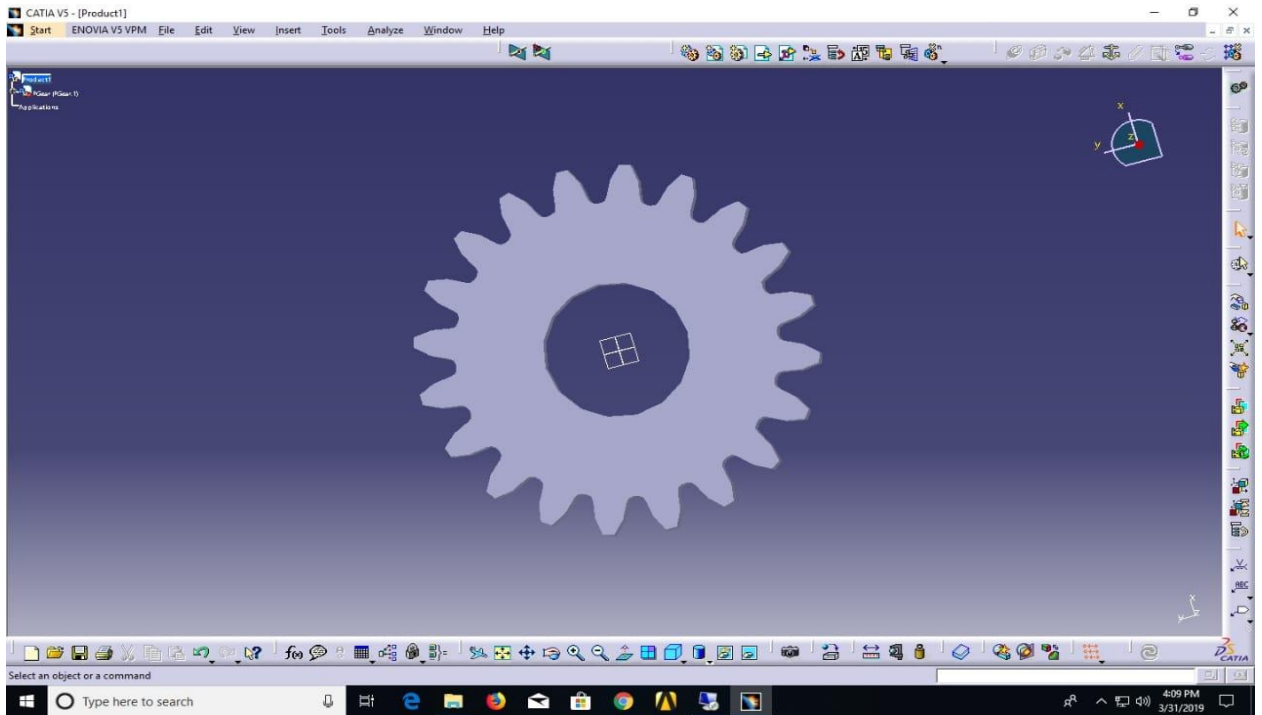
Fig4.6 Part Design of pinion in Catia

4.4 Assembly:

This module is responsible for assembling the parts previously produced in part design, and it is most important for those who work in the field of machinery design or design in general, because it is the one who shows the inter-relationships between the parts of the machine or any mechanical establishment.
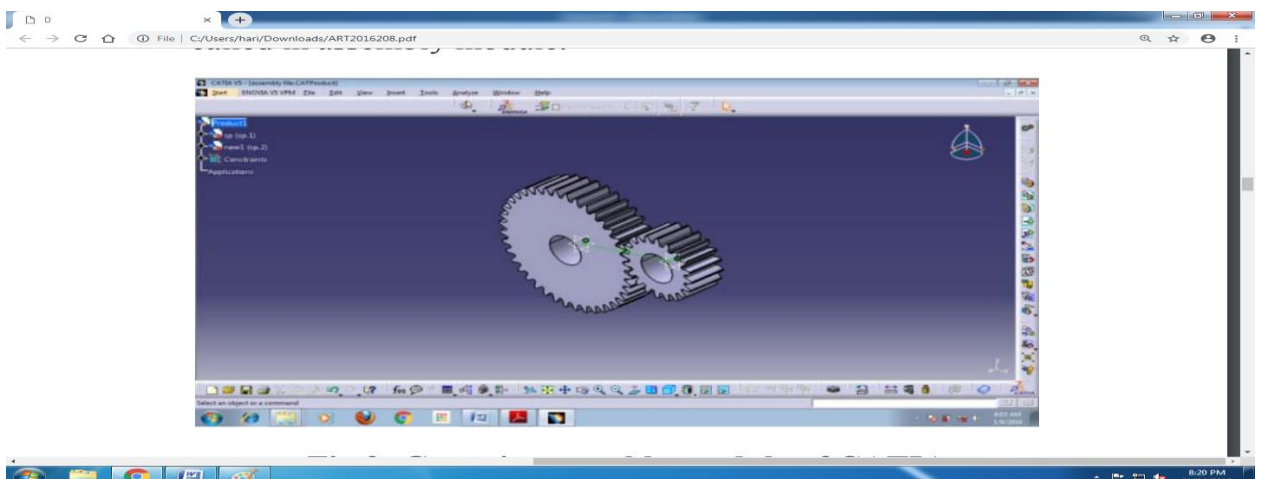
Fig4.7 Parts Assembly in Catia

4.5 Drafting:

This module is responsible, for converting what you see on the screen to standard engineering drawings can be traded in the workshop for manufacturing or save them for documentation.

4.6 Surface and Wire Frame:

With this module surfaces can be drawing with zero size and weight and has its uses in the aerospace, automotive, ships and Mold Design.

4.7 Simulation:

This module is responsible for obtaining a similar movement of the natural movement, which is expected to occur during the actual operation of the machine or mechanical establishment whatever

4.8 Free Style:

Which is a free drawing, product designers needs it, such as Mobile or fumiture or antiques designers and other modules such as

4.8.1 Sheet Metal, Mold Design, Welding, Aerospace Sheet Metal:

The surprise is that all of the above follows the one field which is mechanical design field, while there are other fields such as:

  1. Analysis

  2. Machining

  3. Ergonomics

Each of them containing modules, even electronic circuit design, that is mean CATIA have too many modules, and it is covers almost everything you need, so I prefer to work on it more than others.

CATIA is considered as a CAM program, in addition to it is CAD program, in the meaning that you can export files to CNN machines and then manufactured, CATIA Also supports graphics from other programs such as AutoCAD, for example, it is possible to copy a drawing from AutoCAD and enter it to CATIA and then make on it

CATIA operations, CATIA files can be kept with dwg extension which is supported by AutoCAD or the default extension has.

4.9 DESIGN PROCEDURE FOR SPUR GEAR:

- We define the number of parameters that we need in our sketch, for that open formulas list i.e., f(x)
- The one would be real type parameter and the other would be of length type parameters.
- Create, one parameter of the real type and name the parameter as N (It is for the number of teeth) and give the value.
- Create, another parameter of length type and name the parameter as m (Module) and give the value.
- Make one more parameter of length type and name the parameter as Rp (Radius of Pitch circle) and we have to add a formula for this i.e., Rp=m*N/2. Thus the value is automatically calculated.
- We will define one more parameter of length type and name the parameter as Rb (Radius of Clearance (base) circle) and add a formula i.e., Rb=0.94*Rp. The value is automatically calculated.
- One more new parameter of length type and name the parameter as Ra (Radius of Addendum circle) and add a formula i.e., Ra=Rp+m. The value is automatically calculated.
- The last parameter of length type and name the parameter as Rd (Radius of Dedendum circle) and add a formula i.e., Rd= Rp-1.25*m. The value is automatically calculated.
- All new list of parameters with their values will be applying it to our workbench and we will start with the figure with a yz plane.
- At first, we will make base circle as well as the pitch circle.
- Select a circle command to draw and give the dimension to the circle by double click on it.
- Now we will be applying our parameter Rp to this pitch circle radius so the radius would automatically calculated and adjusted and similarly for base circle radius (Rb) also.

- Now we need to create two construction lines and the angle between this two lines would be 20 deg (as it is a spur gear with a pressure angle of 20 deg)and it can be define at the new parameters of angle type also.

- We will draw one more construction line, which is a vertical line between the above two inclined lines and the angle between the above inclined line and vertical line can be defined by a formula i.e., ta=90/N*1 deg. The value is automatically calculated.

- Now we need to draw a circle to get the particular profile. For drawing the circle, the inclined line and the base circle radius meets at a point on right side by taking it as the centre upto the point where the pitch circle radius and another inclined line meets.

- Make the base and pitch circle radius as construction circles as well as hide them as we don't need them know.

- We will be preceding with remaining two circles i.e., Addendum and Dedendum circle.

- Now we will be applying our parameter Ra addendum circle so the radius would automatically calculated and adjusted and similarly for dedendum circle radius (Rd) also.

- Now we will trim the unwanted parts and hence, we will see the half tooth profile.

- Now we can define a fillet radius also by a formula i.e., fr=0.39*m.

- By using mirror command we can see the full tooth profile.

- We need to select lines as well as fillet curve for which we need to make it circular over the profile.

- Now we will go to circular pattern, will select the complete crown and the instances would be our number of teeth i.e., N and the reference element would be our x-axis and give a preview.

- We will use a pad command with a mirror extent of length 10mm each side.

- We need to create a hole for the shaft for that select y-z plane and draw a circle at the centre and use a pocket command with a mirror extended of 10mm each side.

- So we have completed the spur gear of number of teeth 20 and the module of 3.

- Similarly, same for the part design of pinion with different values.

- Assemble the parts and coincide.

CHAPTER 5

FINITE ELEMENT ANALYSIS

5.1 Introduction to finite element analysis:

Approximate solutions to a wide variety of engineering problems although originally developed to study stresses in complex airframe structures, it has since been extended and applied to the broad field of continuum mechanics. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering schools and industry. The finite element method has become a powerful tool for the numerical solution of a wide range of engineering problems. Advances in computer technology and CAD systems, has led to increased use of FEM in research as well as industry as complex problems can be modeled and released with relative ease.

5.2 Basic Steps in the Finite Element Analysis:

The basic steps involved in finite element analysis consist of the following

a) **Pre-Processing Phase:**

Create and discretize the solution domain into finite elements i.e. subdivide the problem into nodes and elements.
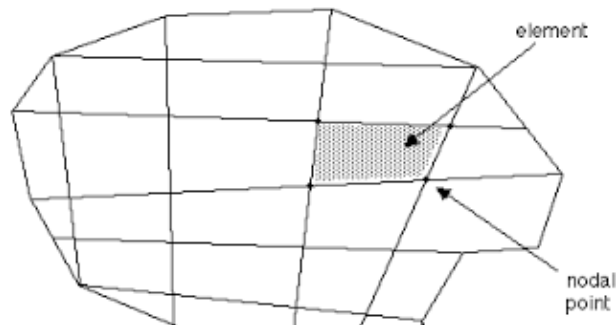
Fig 5.1 Discretization of Real Continuum with Node and Element

- Assume a shape function to represent the physical behavior of an element; that is an approximate continuous function is assumed to represent the solution of an element.

- Develop equations for all the elements in the mesh.

- These generally take form

31

- $[K]\{U\} = \{F\}$
- Where '$[K]$' is a square matrix, known as stiffness matrix
- '$\{U\}$' is the vector of (unknown) nodal displacements or temperature
- '$\{F\}$' is the vector of applied nodal forces
- Assemble the elemental equations to obtain the equations of the whole problem. Construct the global stiffness matrix.
- Apply boundary conditions, initial conditions, and loading.

  b) **Solution Phase:**

  Solve a set of linear or nonlinear algebraic equations simultaneously to obtain nodal results of primary degrees of freedom or unknowns, such as displacement values at different nodes in structural problem or temperature values at different nodes in heat transfer problem.

  c) **Post processing phase:**
- Computation of any secondary unknowns or variables e.g. the gradient of the solution.
- Interpretation of the results to check whether the solution makes sense.
- Tabular and/or graphical presentation of the results.
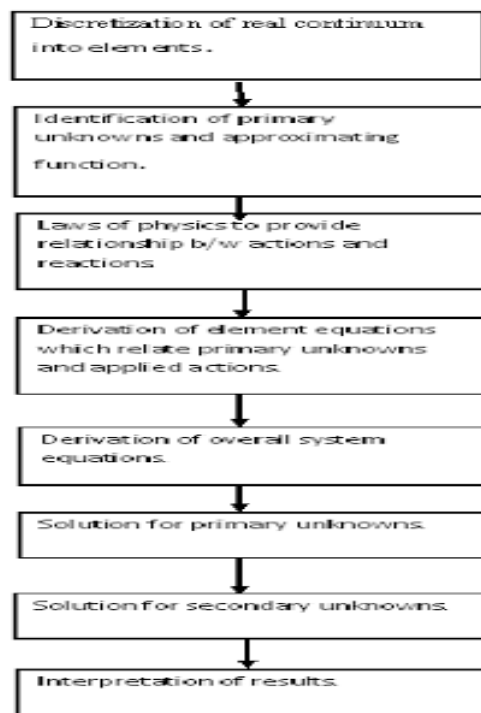
Fig 5.2 Tabular presentation of results

5.3. **Introduction to ANSYS Workbench:**

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 analysis can be performed with drag-and-drop simplicity.
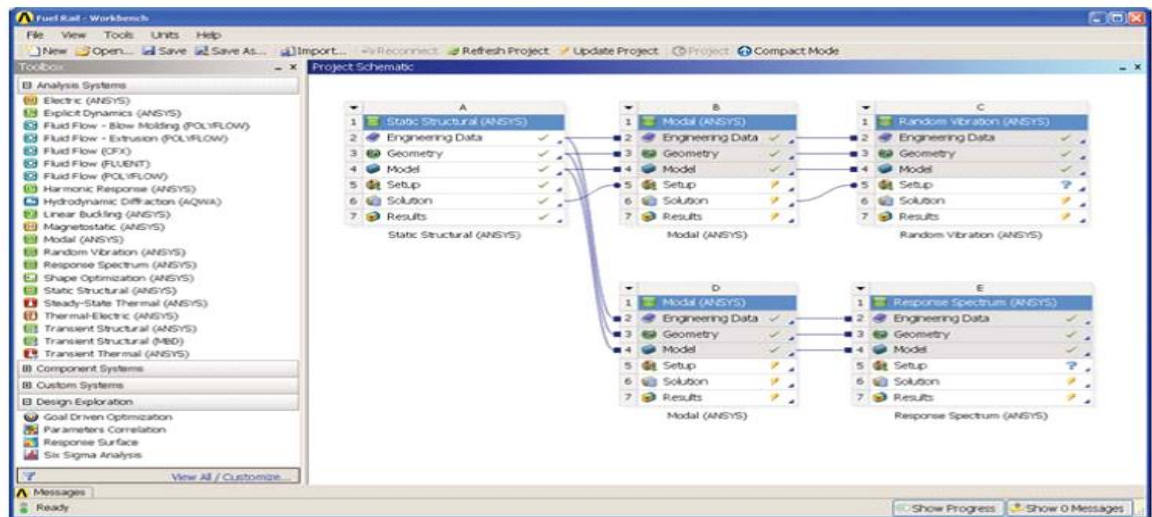
Fig 5.3 Workbench 19.2 Interface

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 analyses. 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 analyses 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.

5.3.1. ANSYS Workbench Features:

- Bidirectional, parametric links with all major CAD systems.
- Integrated, analysis-focused geometry modeling, 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.
- Complex project schematics can be saved for re-use.
- Pervasive, project-level parameter management across all physics.
- Automated what-if analyses with integrated design point capability.
- Adaptive architecture with scripting and journaling capabilities and API's enabling rapid integration of new and third-party solutions.

5.4. Structural Analysis:

Structural analysis is probably the most common application of the finite element method. The term *structural* (or *structure)* implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

5.4.1. Types of structural analyses:

The seven types of structural analyses available in the ANSYS family of products are explained below. The primary unknowns (nodal degrees of freedom) calculated in a structural analysis are displacements. Other quantities, such as strains, stresses, and reaction forces, are then derived from the nodal displacements.

Structural analyses are available in the ANSYS/Multi physics, ANSYS/Mechanical, ANSYS/Structural, and ANSYS/Linear Plus programs only.

One can perform the following types of structural analyses

1) Static Analysis:

Used to determine displacements, stresses, etc., under static loading conditions. It comprises of both linear and non-linear static analysis. Non-linearity can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

2) Model Analysis:

Used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available.

3) Harmonic Analysis:

Used to determine the response of a structure to harmonically time-varying loads.

4) Transient Dynamic Analysis:

Used to determine the response of a structure to arbitrarily time-varying loads. All non- linearity mentioned under Static Analysis above are allowed.

5) Spectrum Analysis:

An extension of the modal analysis, used to calculate stresses and strains due to a response spectrum or a PSD input (random vibrations).

6) Buckling Analysis:

Used to calculate the buckling loads and determine the buckling mode shape.Both linear(Eigen value) buckling and nonlinear buckling analyses are possible.

7) Explicit Dynamics Analysis:

ANSYS provides an interface to the LS-DYNA explicit finite element program and is used to calculate fast solutions for large deformation dynamics and complex contact problems.

- In addition to the above analysis types, several special-purpose features are available

- Fracture mechanics

- Composites

- Fatigue

- p-Method

5.5. Thermal Analysis:

A thermal analysis calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities of interest are

- The temperature distributions

- The amount of heat lost or gained

- Thermal gradients

- Thermal fluxes

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses (that is, stresses caused by thermal expansions or contractions).

Only the ANSYS/Multi physics, ANSYS/Mechanical, ANSYS/Thermal, and ANSYS/FLOTRAN programs support thermal analysis.

The basis for thermal analysis in ANSYS is a heat balance equation obtained from the principle of conservation of energy. The finite element solution one performs via ANSYS calculates nodal temperatures and then uses the nodal temperatures to obtain other thermal quantities.

The ANSYS program handles all three primary modes of heat transfer: conduction, convection, and radiation.

5.5.1 Types of thermal analysis

ANSYS supports two types of thermal analysis

   I.   *A* **steady-state thermal analysis** determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.

   II.   *A* **transient thermal analysis** determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

5.6 PROCEDURE FOR PERFORMING STATIC STRUCTURAL ANALYSIS:

STEP 1: Engineering Data

The data to be calculated is to be submitted in this module. Properties such as yield strength, young's modulus, poisson's ratio, factor of safety are to be provided.
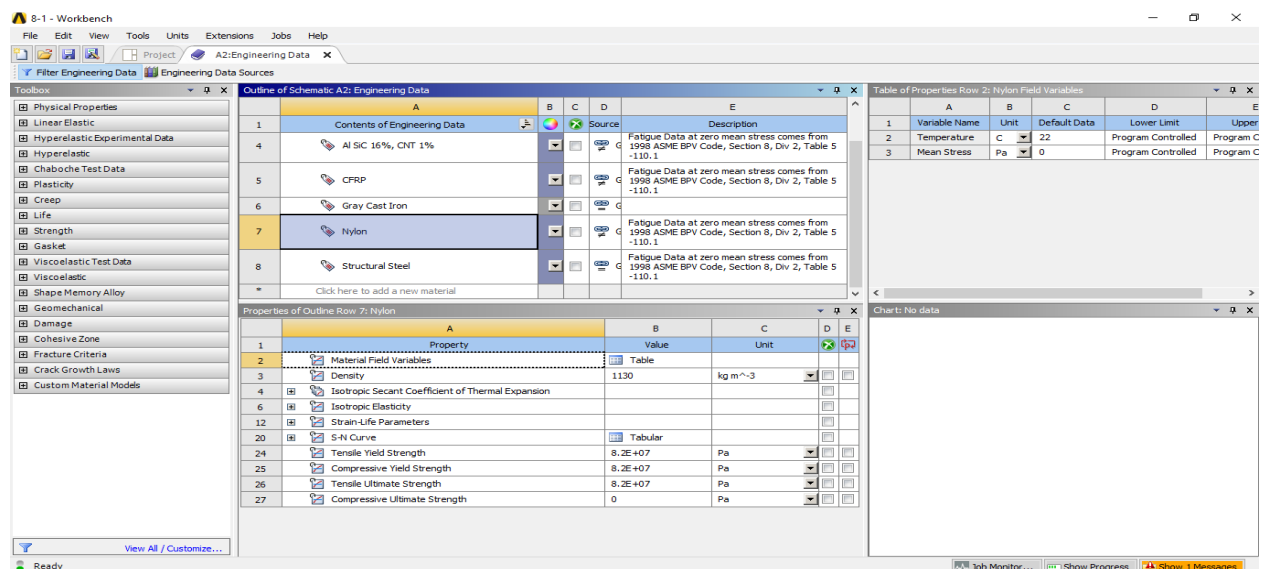
Fig 5.4 ANSYS interface
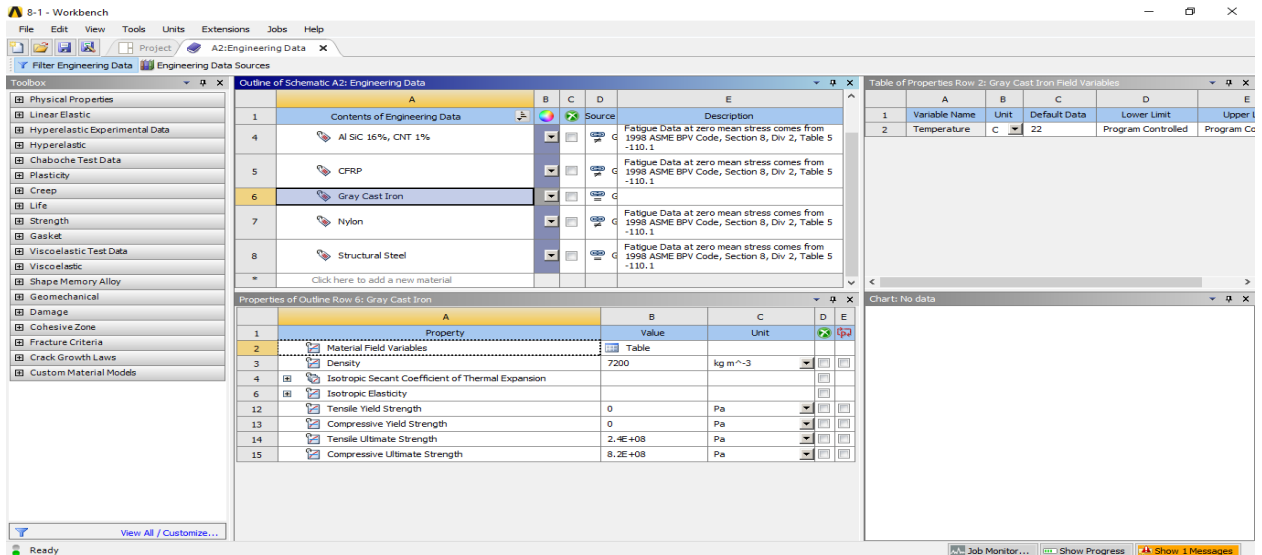
Fig 5.5 Insertion of data in ANSYS

STEP 2: INSERTION OF GEOMETRY

Right click on the geometry and then click on **Import Geometry**. Then close the present tab and again right click on the geometry then click on the **Modify Designer Tool**.

STEP 3: CONSTRAINTS

Right click on the model and apply the constraints on the gear and then apply the loads.

STEP 4: MESHING

Click on mesh option and extend the sizing option then the required data is given as follows:

Size function – Curvature

Minimum size -0.5mm

Maximum face size-1mm

Maximum Tet size-1mm

Growth rate-1.2mm

Deformation size-0.5mm.

Then right click on mesh and click on generate mesh. Then automatic mesh is generated over the model.
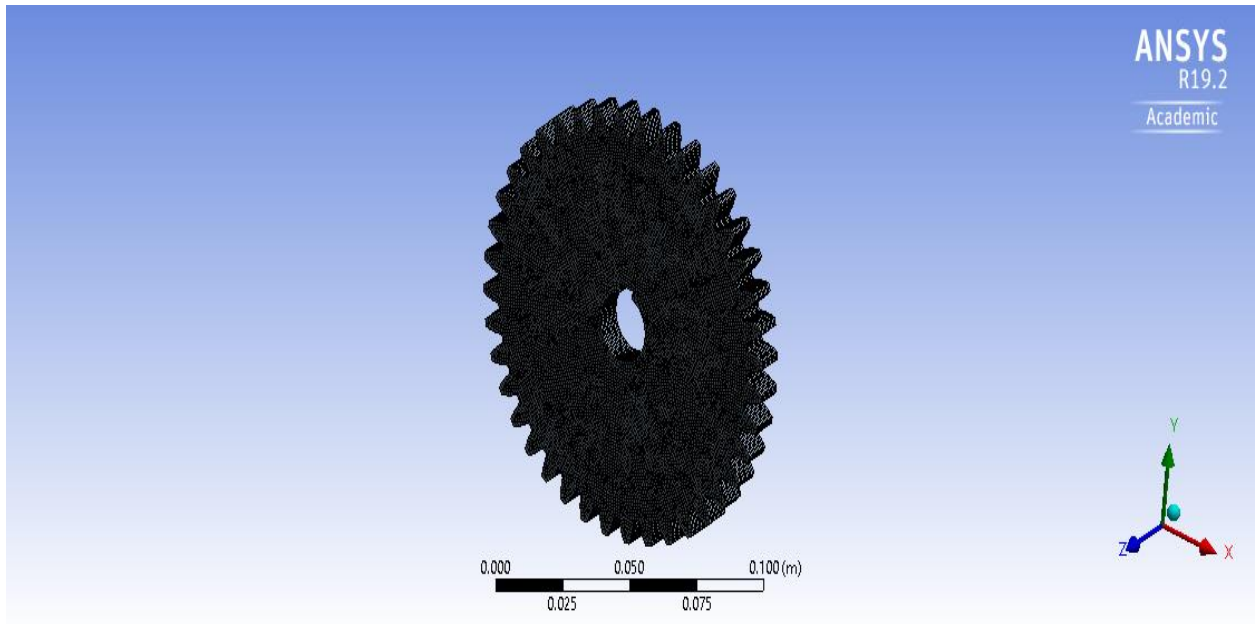
Fig 5.6 Meshing

STEP 5: INSERTION OF SUPPORTS AND FORCES

Right click on **Static structural** icon. Click on **INSERT**. Then select fixed support and click on apply.

Again right click on **Static structural** icon. Click on **INSERT.** Then select force and click on apply. Then the force is defined by the component i.e. in which direction the force is to be applied.

.
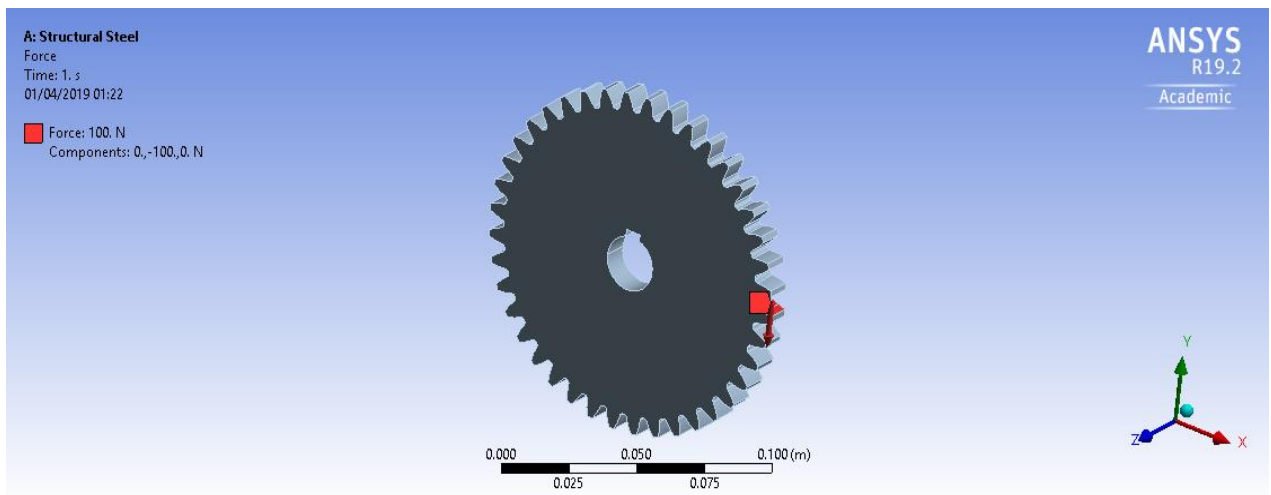
Fig 5.7 Application of load on spur gear

STEP 6: INSERTION OF COMPONENTS WHOSE VALUES ARE TO BE DETERMINED

Click on **SOLUTION** and click on **INSERT** and **STRESS** and then **EQUIVALENT STRESS.**

Similarly **DEFORMATION** is also inserted.

STEP 7: SOLUTION

Click on **SOLVE** on theme menu bar and wait till the results are obtained.

5.7 PROCEDURE FOR PERFORMING THERMAL ANALYSIS

Similar procedure is done as in static structural analysis up to **STEP4.**

STEP 5: CONSTRAINTS

Right click on static structural and click on **INSERT** and insert **CONVECTION** and **HEAT FLUX**.

I. Convection is determined by selecting all the faces and deselecting the contact surfaces.
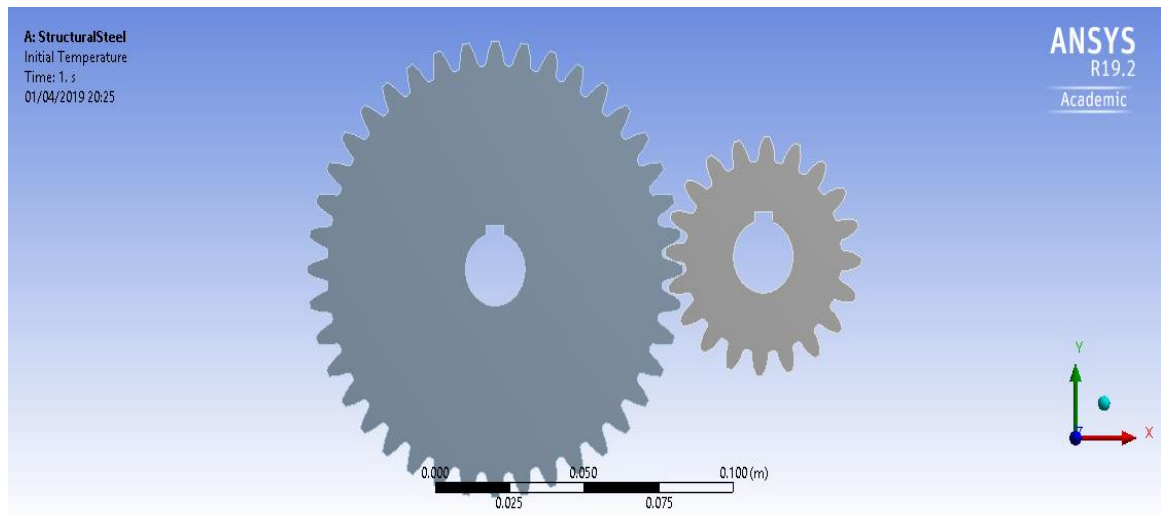
Fig 5.8 Modification of gear in ANSYS workbench
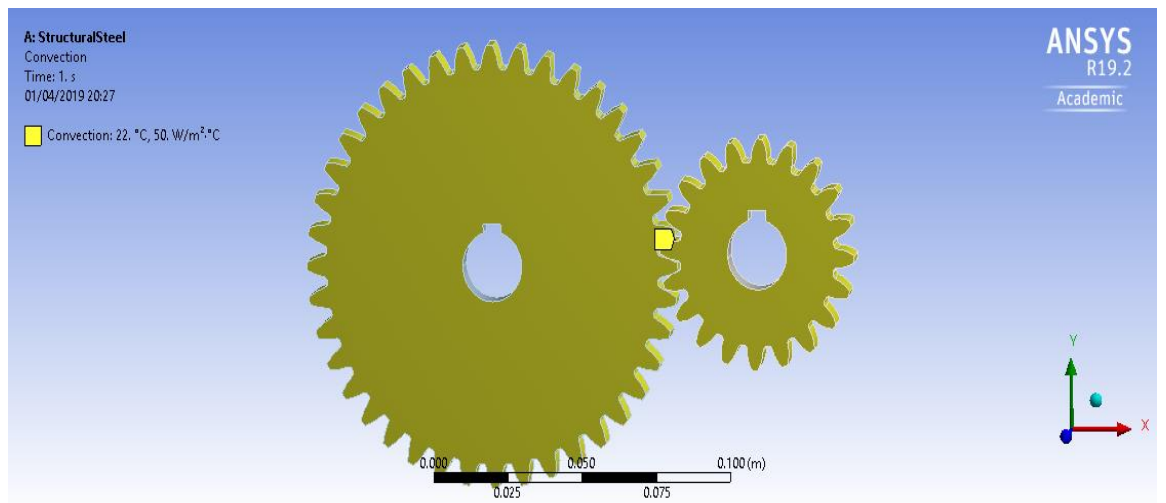
Fig 5.9 Convection of spurgear

STEP 6: INSERTION OF COMPONENTS WHOSE VALUES ARE TO BE DETERMINED

Click on **SOLUTION** and then click on **INSERT** then **THERMAL** and then insert **TEMPERATURE** and **HEAT FLUX.**

STEP 7: SOLUTION

Click on **SOLVE** and the results are obtained.

NOTE: Heat flux is to be found at contact surfaces.

5.8 PROCEDURE FOR PERFORMING CONTACT STRESS ANALYSIS

Similar procedure is done as in static structural analysis up to **STEP4.**

STEP 5: CONSTRAINTS

INSERT **MOMENT, FRICTIONLESS SUPPORTS** after adding constraints.

STEP 6: INSERTION OF COMPONENTS WHOSE VALUES ARE TO BE DETERMINED

INSERT Stresses while considering only contact faces.

STEP 7: SOLUTION

Click on solve and the results are calibrated.

CHAPTER 6

RESULTS AND DISCUSSIONS

6.1 STATIC STRUCTURAL ANALYSIS OF SPUR GEAR AT DIFFERENT LOADS

The analysis is done on various materials and the stress and deformation results will be presented below followed by a comparison between all the used materials using a graphical approach.

6.1.1 For Structural Steel

Equivalent Stresses
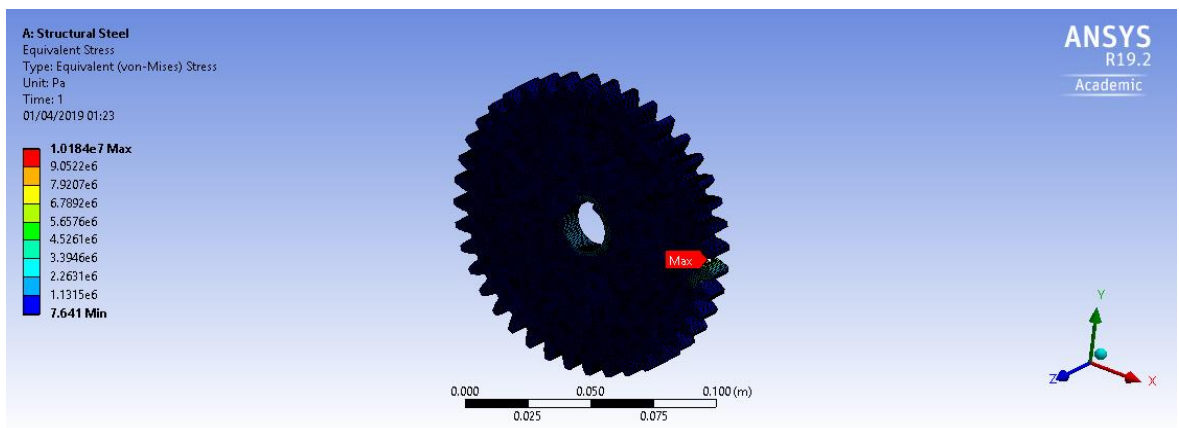
Fig6.1 Equivalent stresses of structural steel in static loading
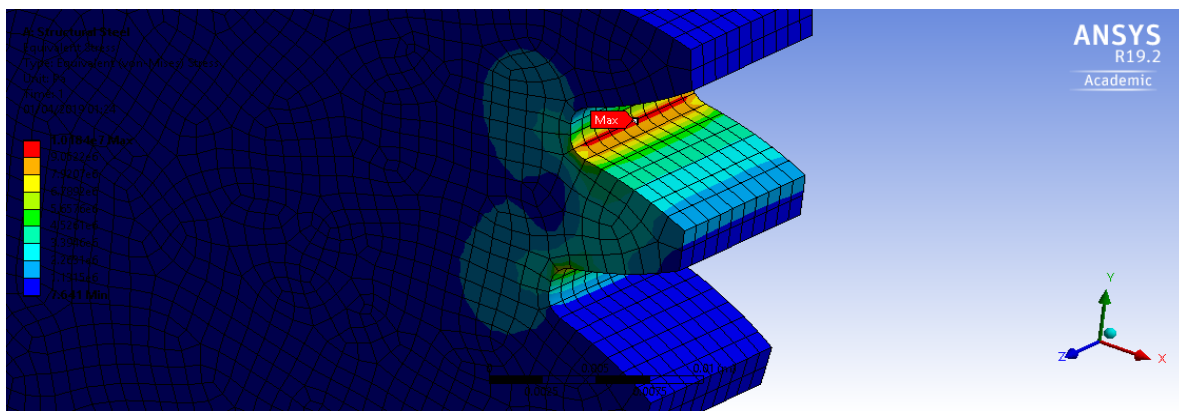
Fig 6.2 Maximum stress representation of structural steel

43
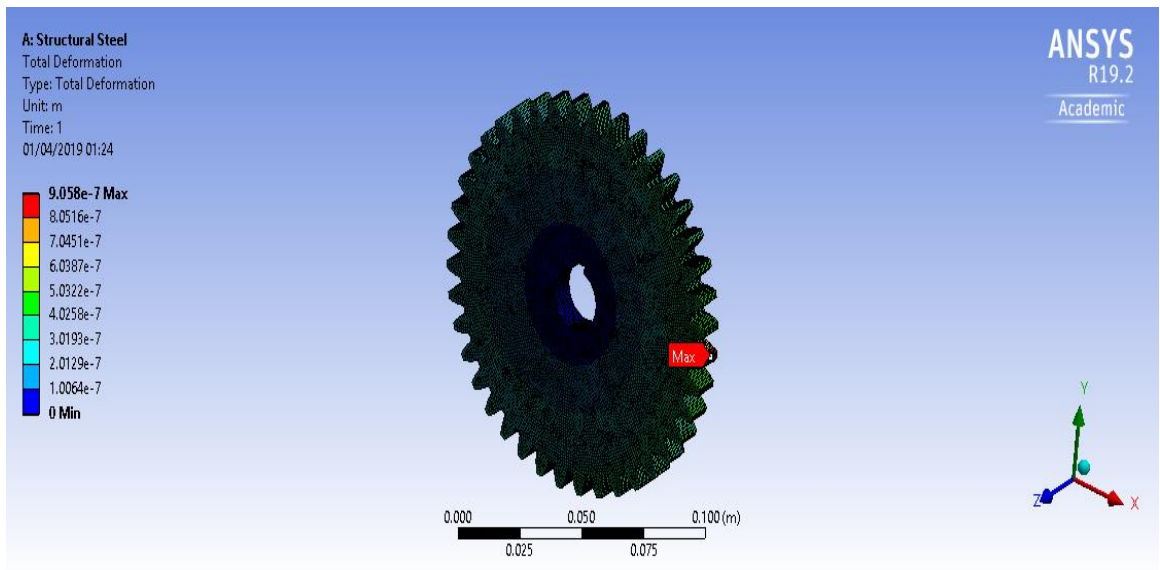
Total Deformation

Fig6.3 Deformation of structural steel in static loading
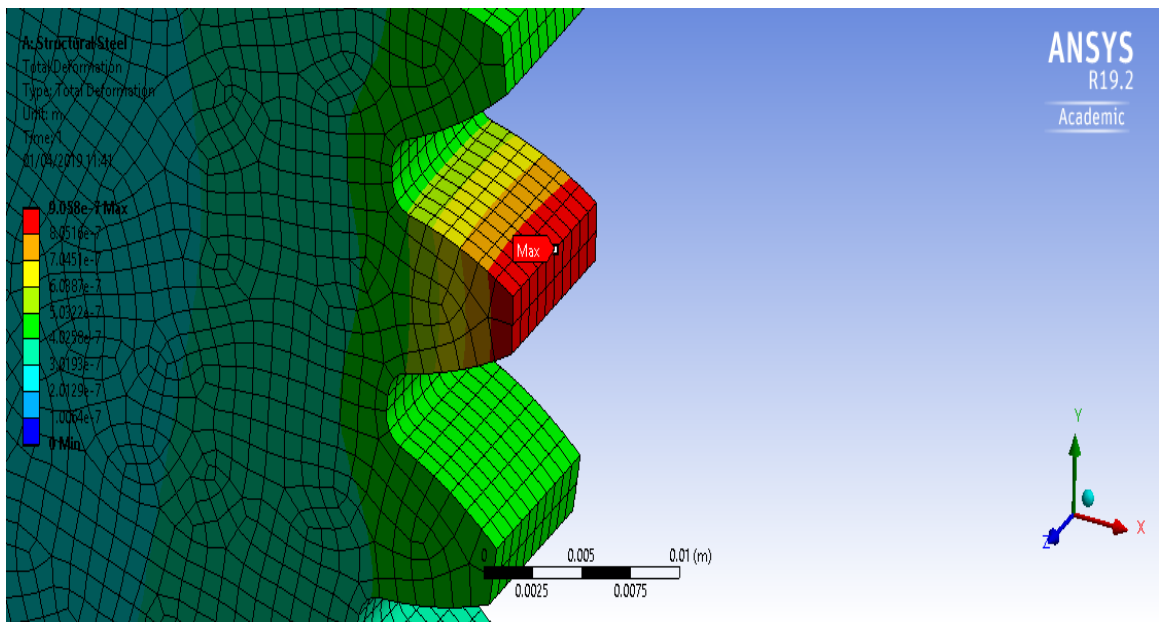
Fig6.4 Maximum deformation representation of structural steel

6.1.2 For Grey Cast Iron

Equivalent Stresses
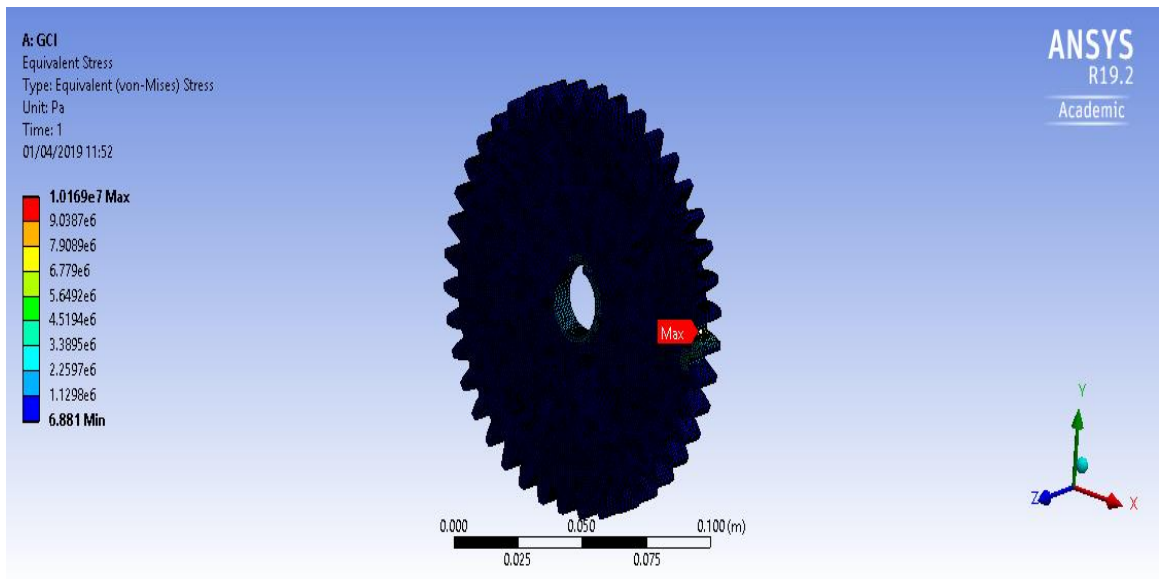
Fig 6.5 Equivalent stresses of grey cast iron in static loading
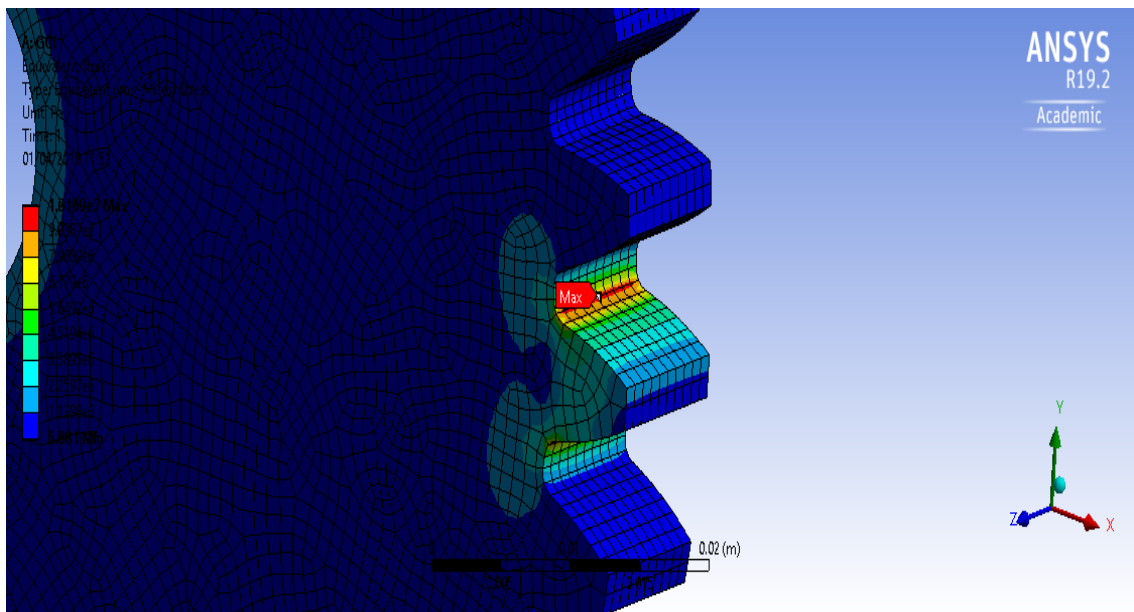
Fig 6.6 Maximum stress representation of grey cast iron

Total Deformation
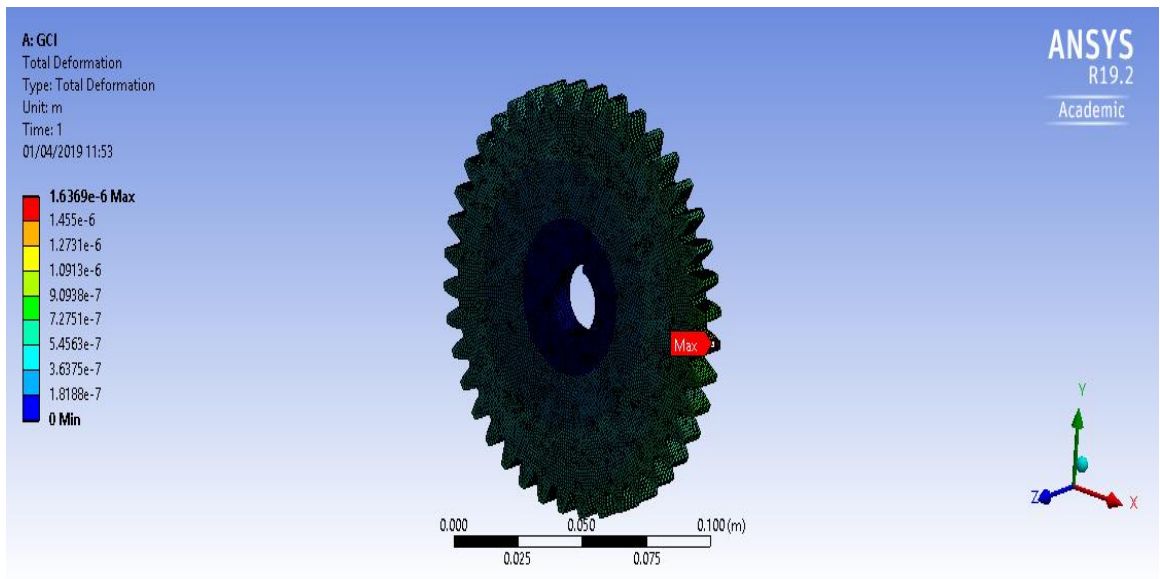
Fig 6.7 Deformation of grey cast iron in static loading
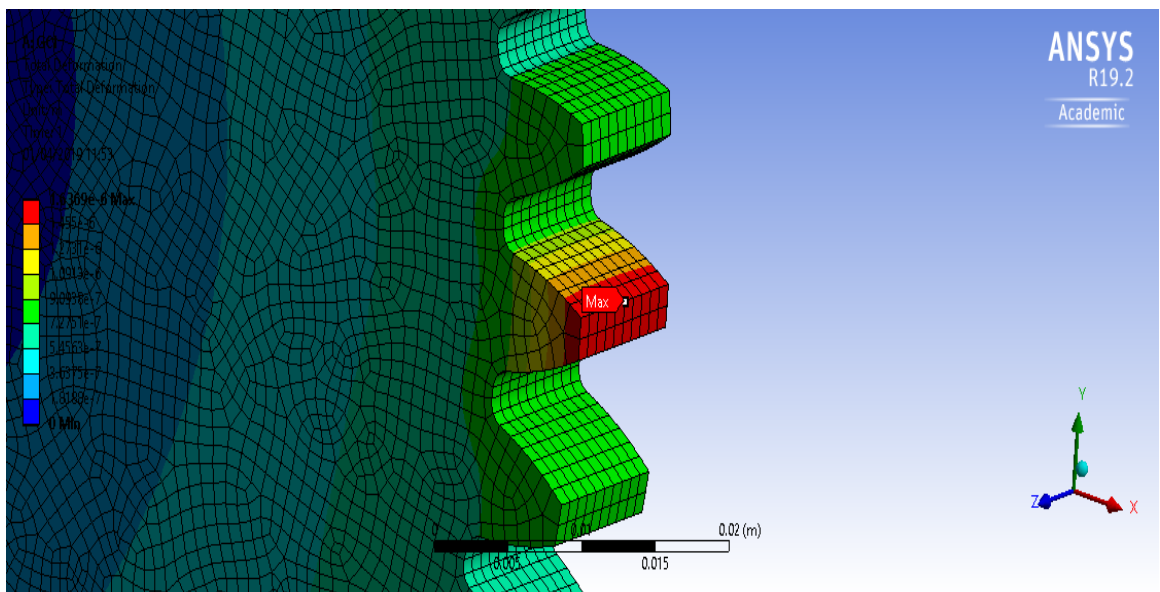
Fig 6.8 Maximum deformation representation of grey cast iron

46

6.1.3 For CFRP

Equivalent Stresses
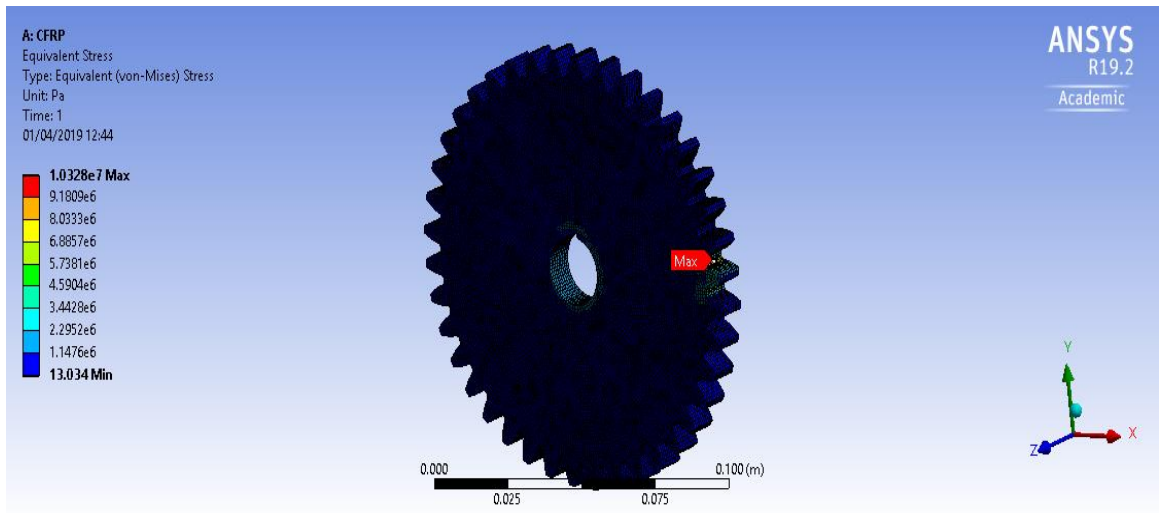
Fig 6.9 Equivalent stresses of CFRP in static loading
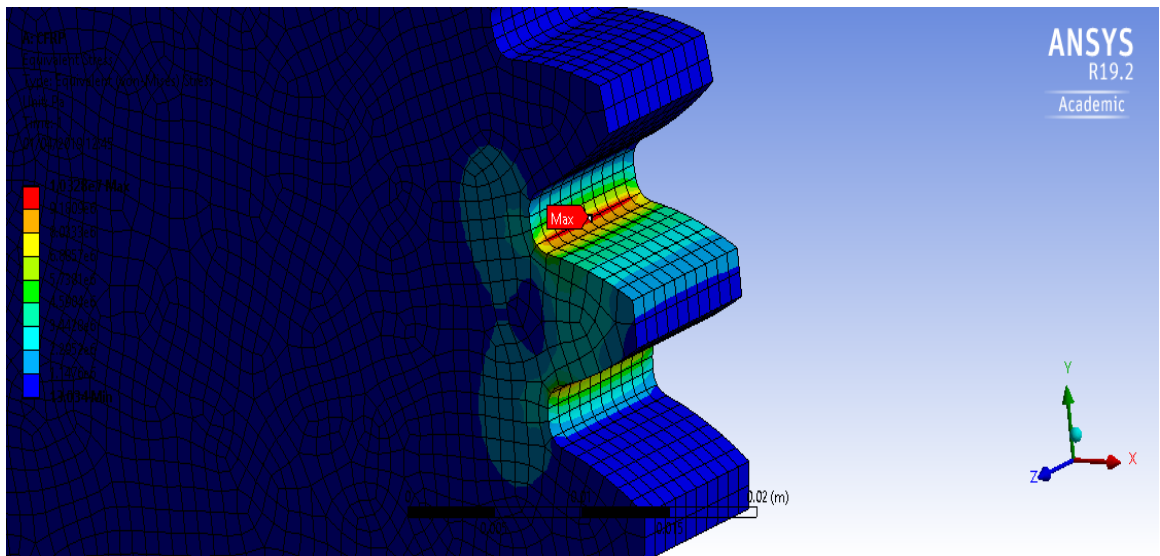
Fig6.10 Maximum stress representation of CFRP

Total Deformation
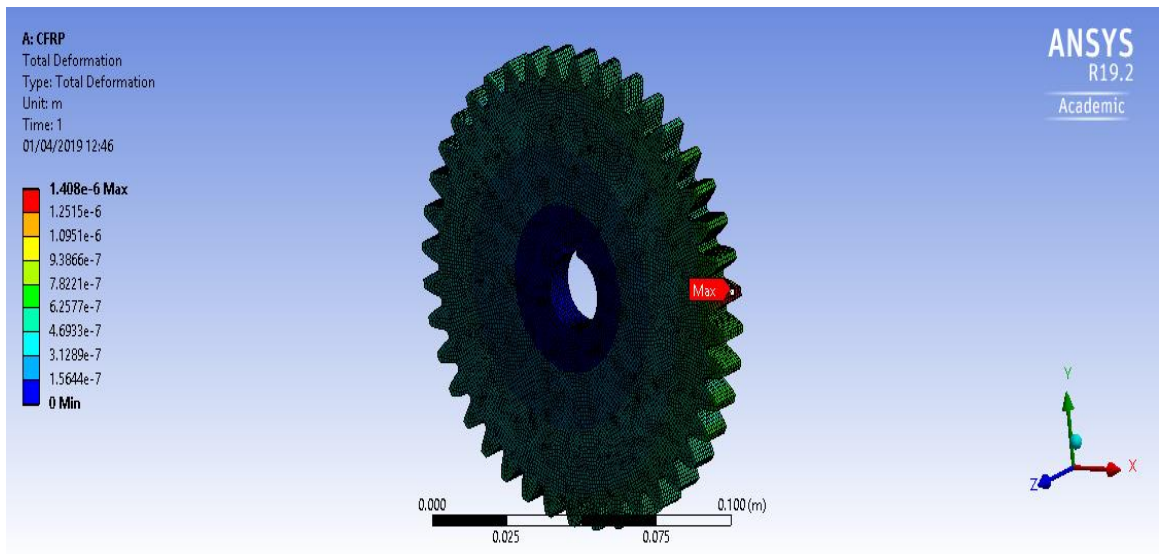
Fig 6.11 Deformation of CFRP in static loading
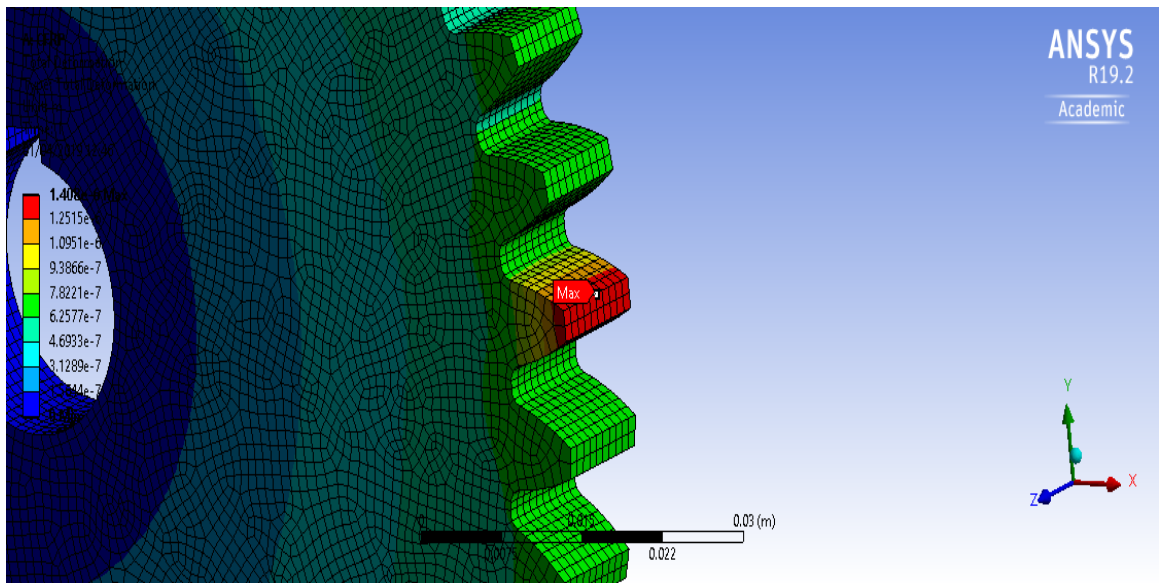
Fig 6.12 Maximum deformation representation of CFRP

6.1.4 For Al SiC 15%

Equivalent Stresses
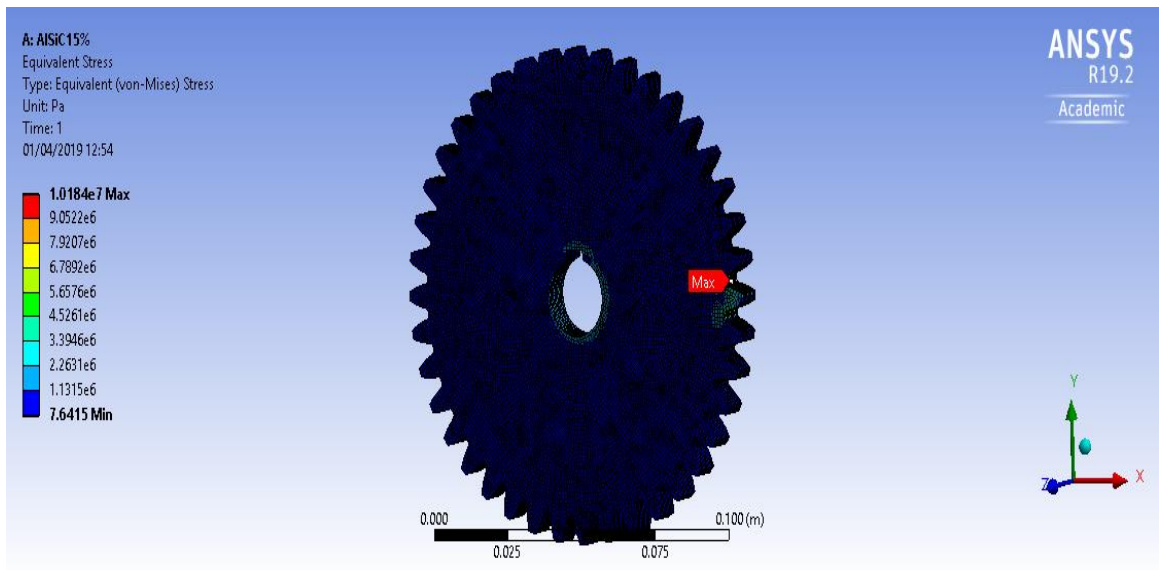
Fig 6.13 Equivalent stresses of Al SiC 15% in static loading
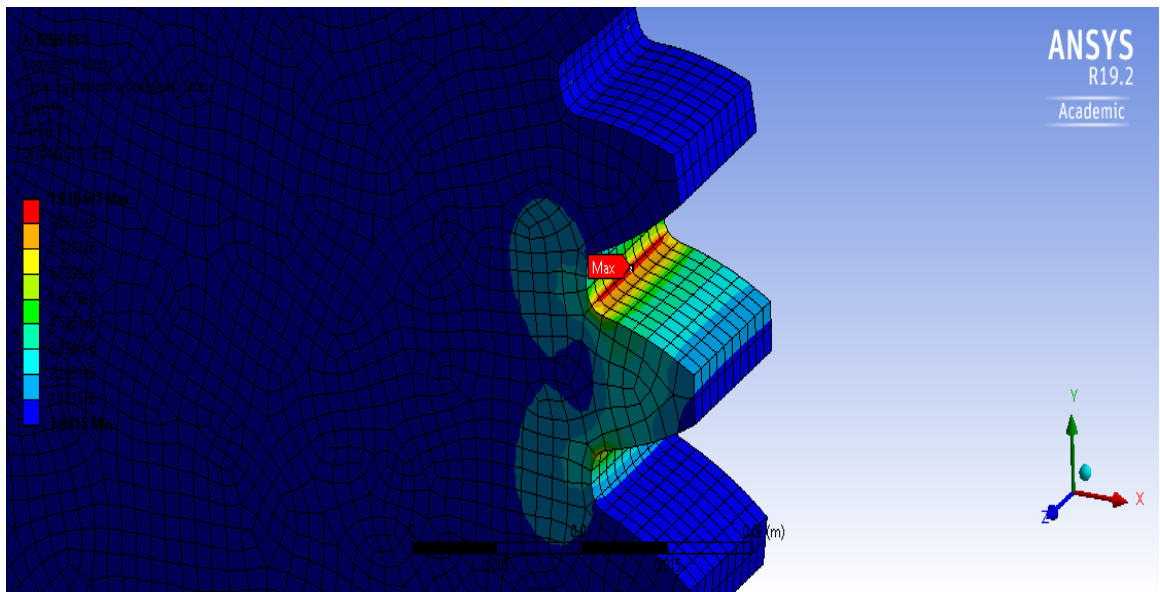
Fig 6.14 Maximum stress representation of Al SiC 15%

Total Deformation
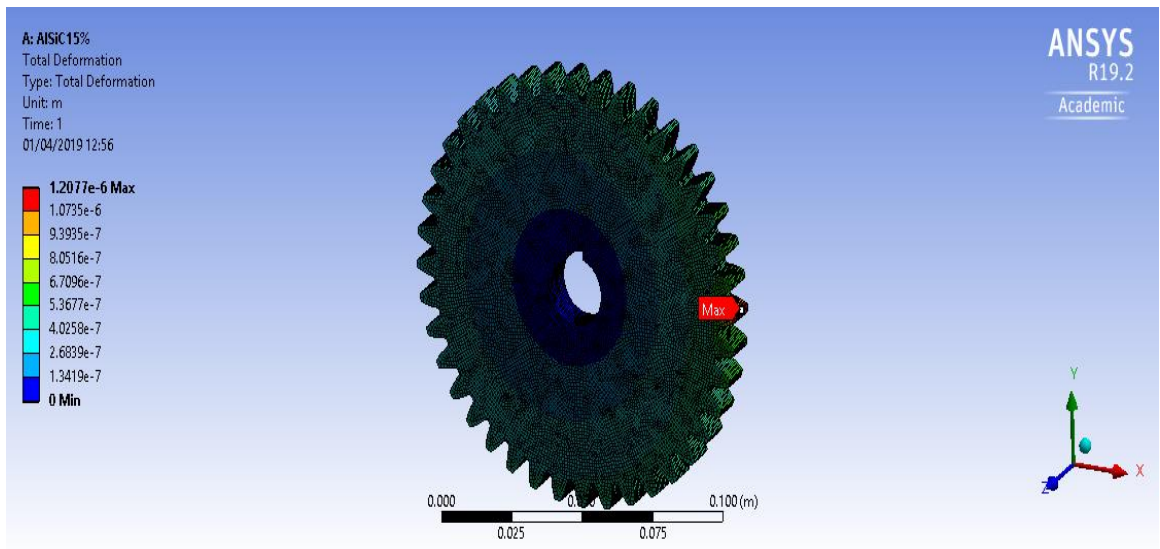
Fig 6.15 Deformation of Al SiC 15% in static loading
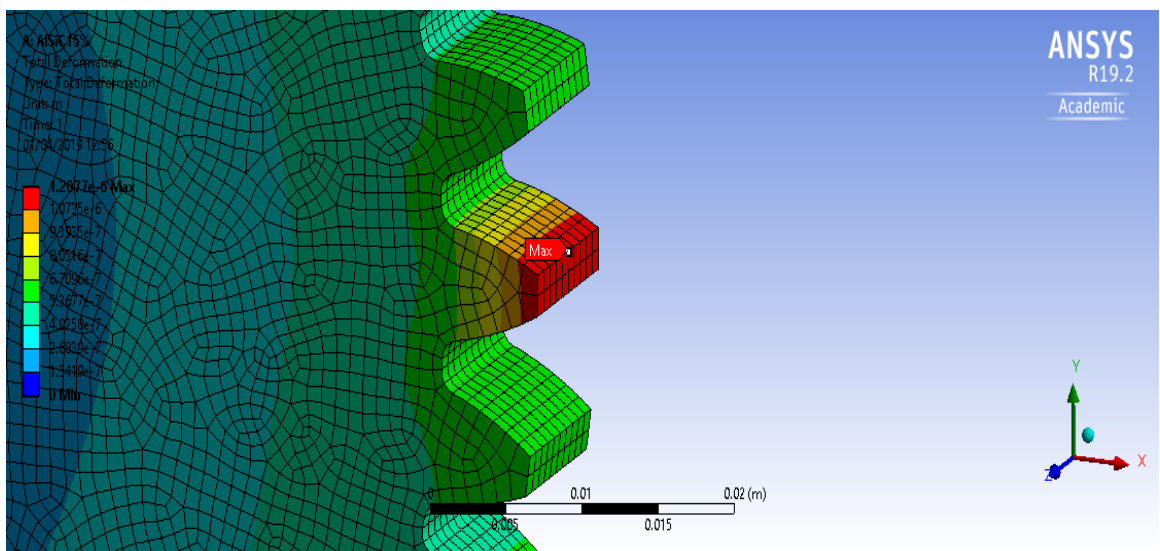
Fig 6.16 Maximum deformation representation of Al SiC 15%

6.1.5 For Al SiC 16%, CNT 1%

Equivalent Stresses
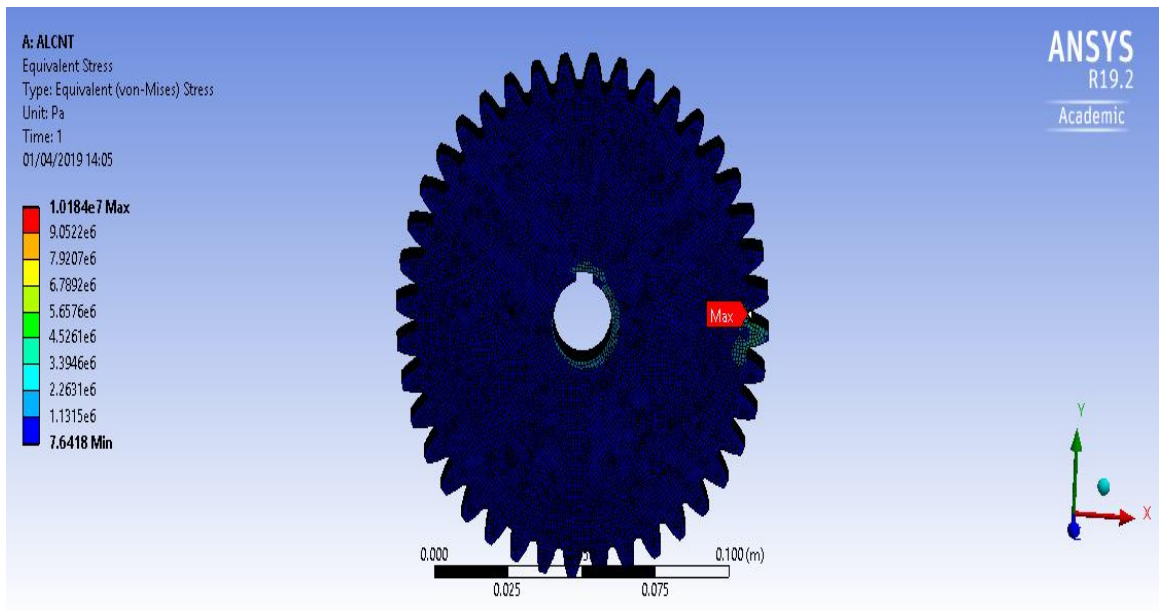
Fig 6.17 Equivalent stresses of Al SiC 16% CNT 1% in static loading
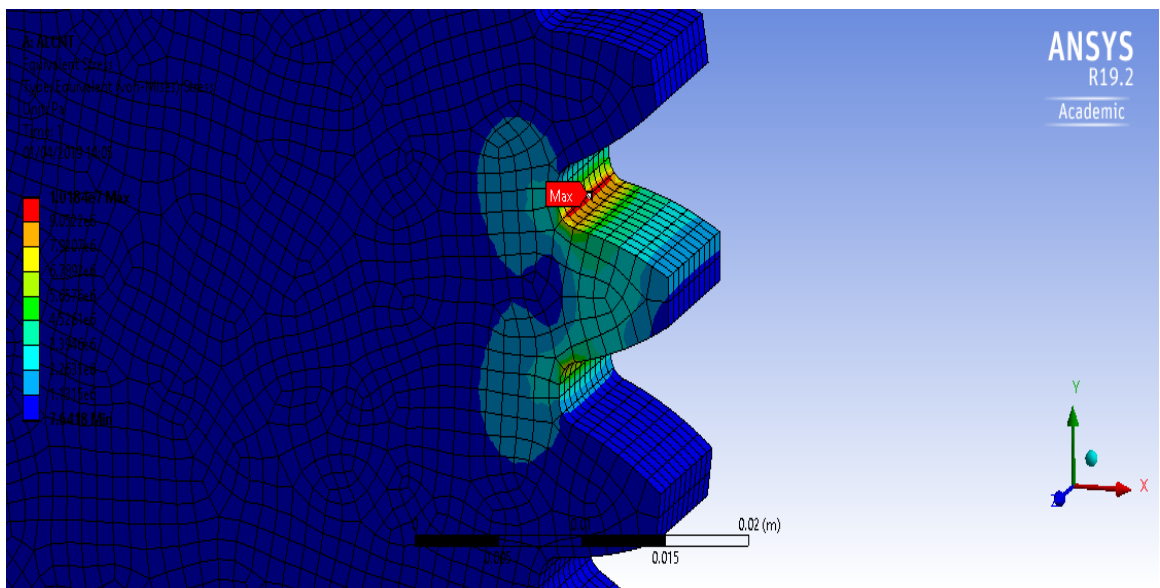
Fig 6.18 Maximum stress representation of Al SiC 16% CNT 1%

Total Deformation
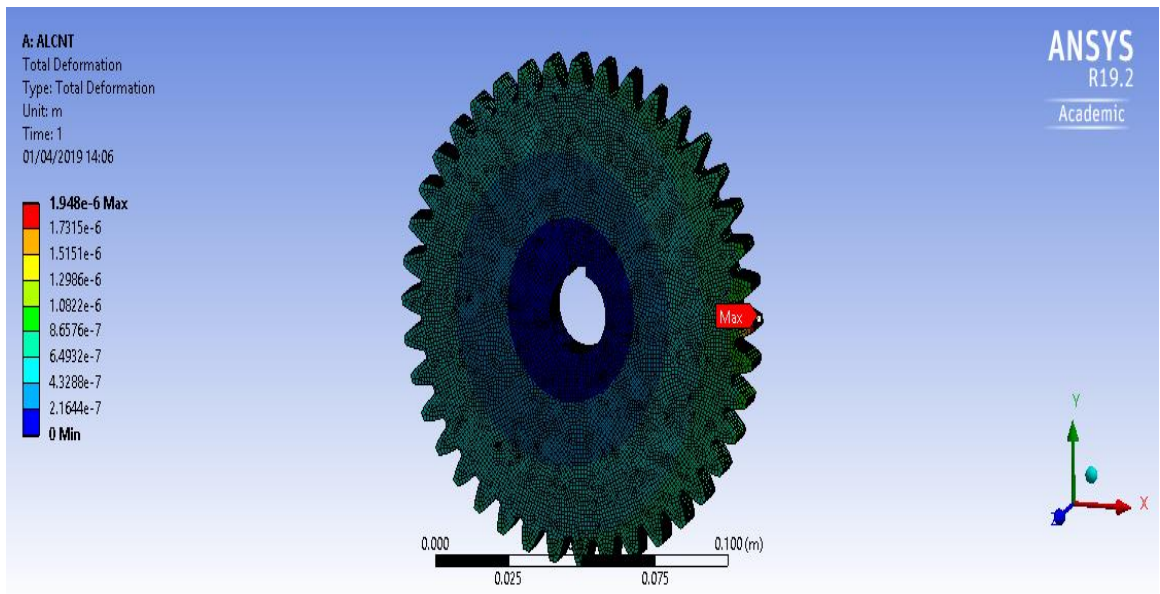
Fig 6.19 Deformation of Al SiC 16% CNT 1% in static loading
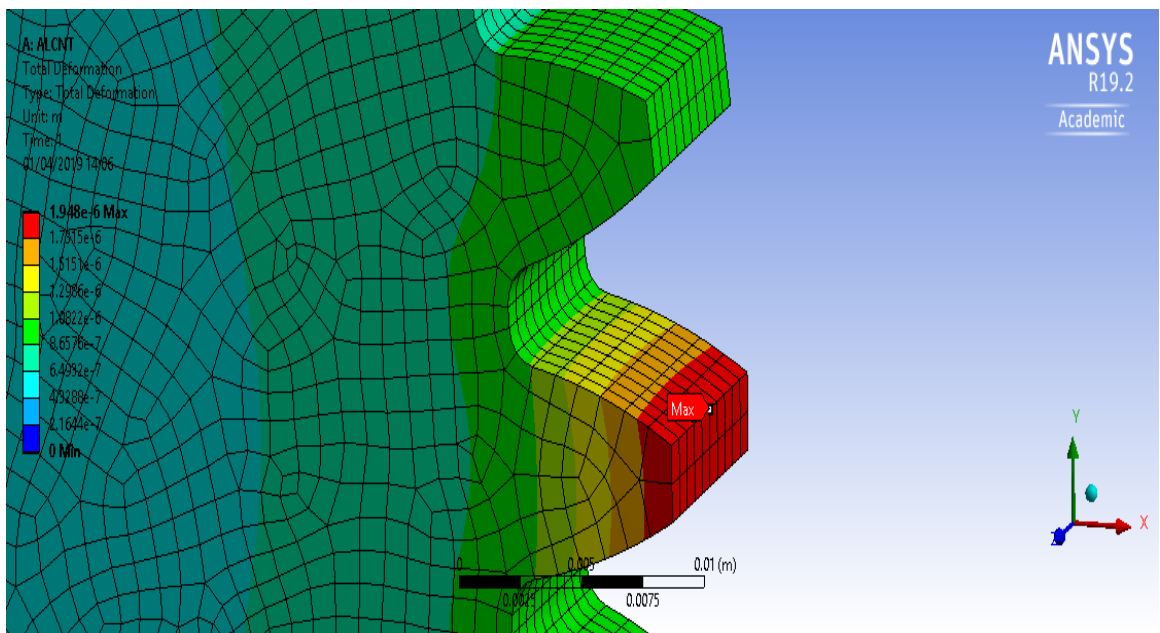
Fig 6.20 Maximum deformation representation of Al SiC 16% CNT 1%

6.1.6 For Nylon

Equivalent Stresses
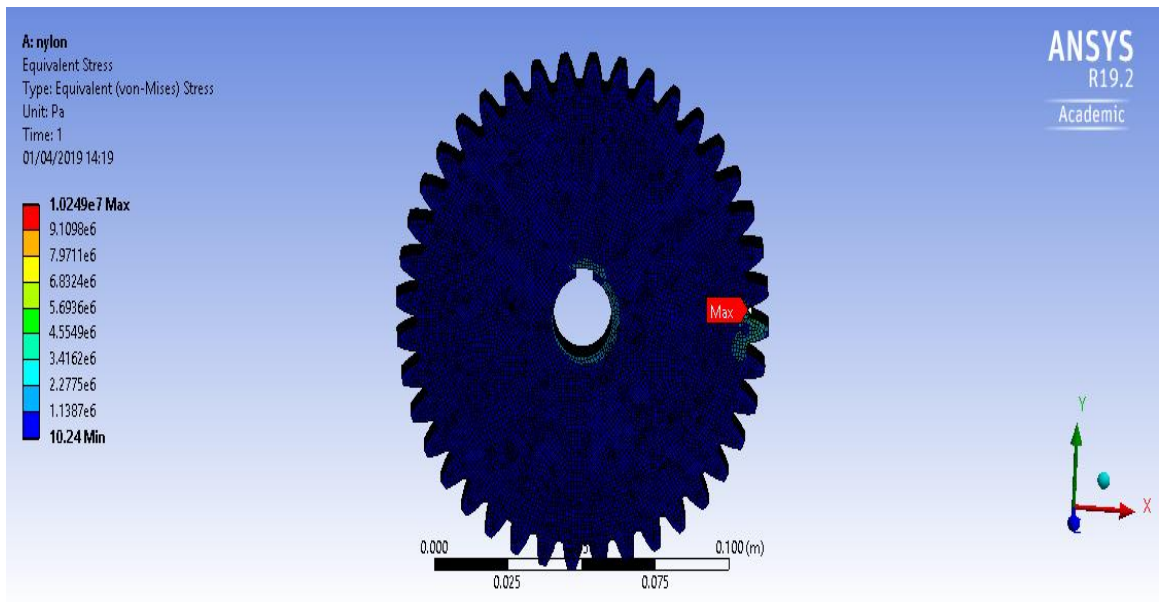
Fig 6.21 Equivalent stresses of Nylon in static loading
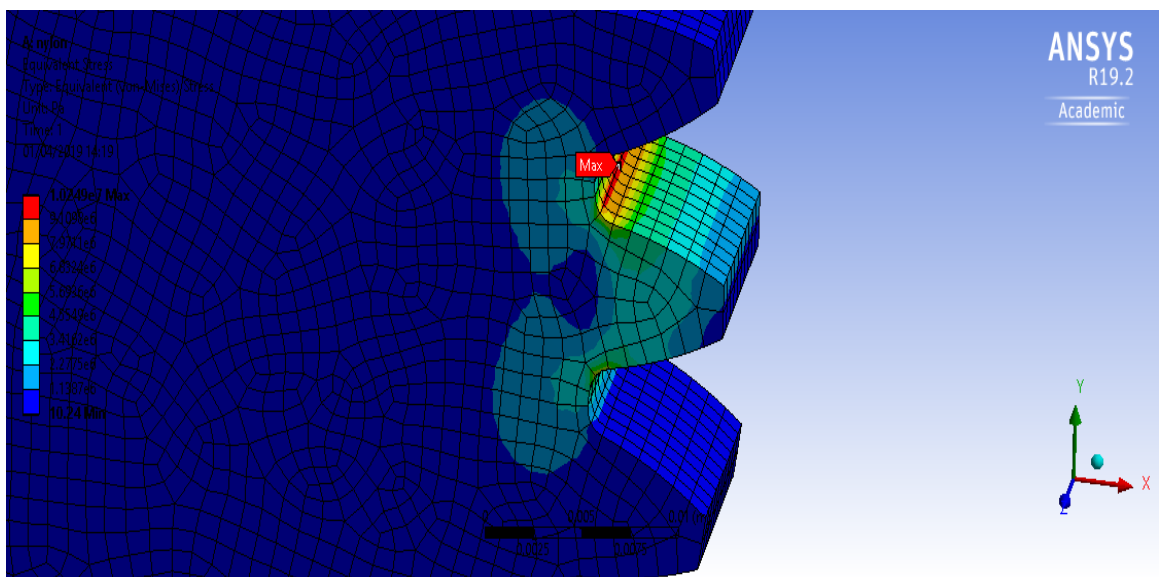
Fig 6.22 Maximum stress representation of Nylon

53
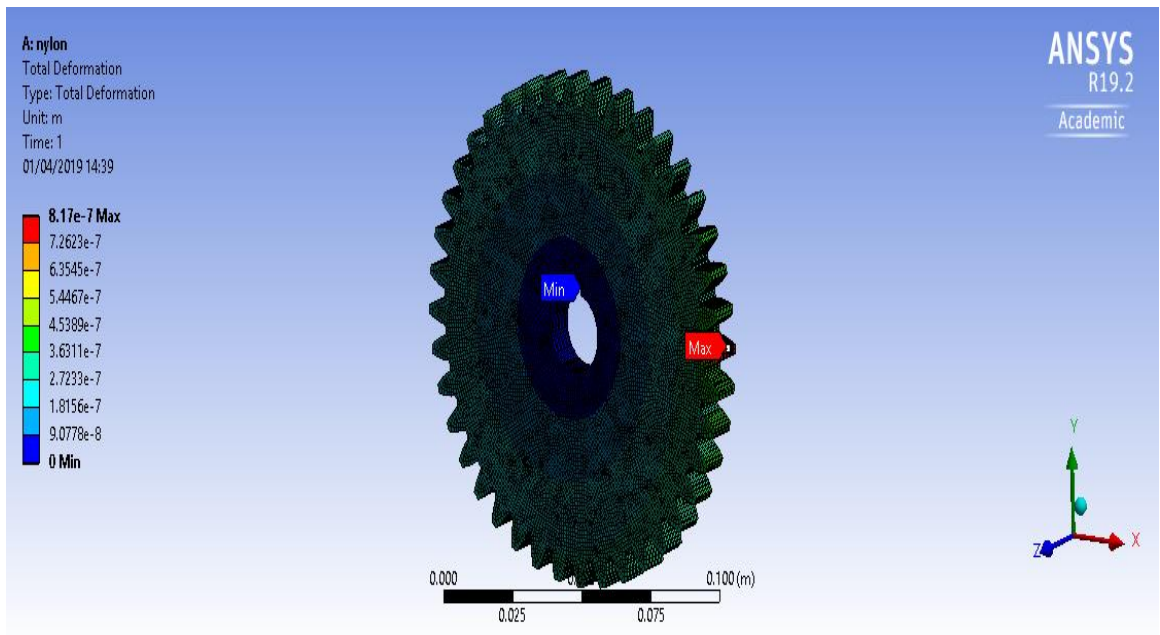
Total Deformation

Fig 6.23 Deformation of Nylon in static loading
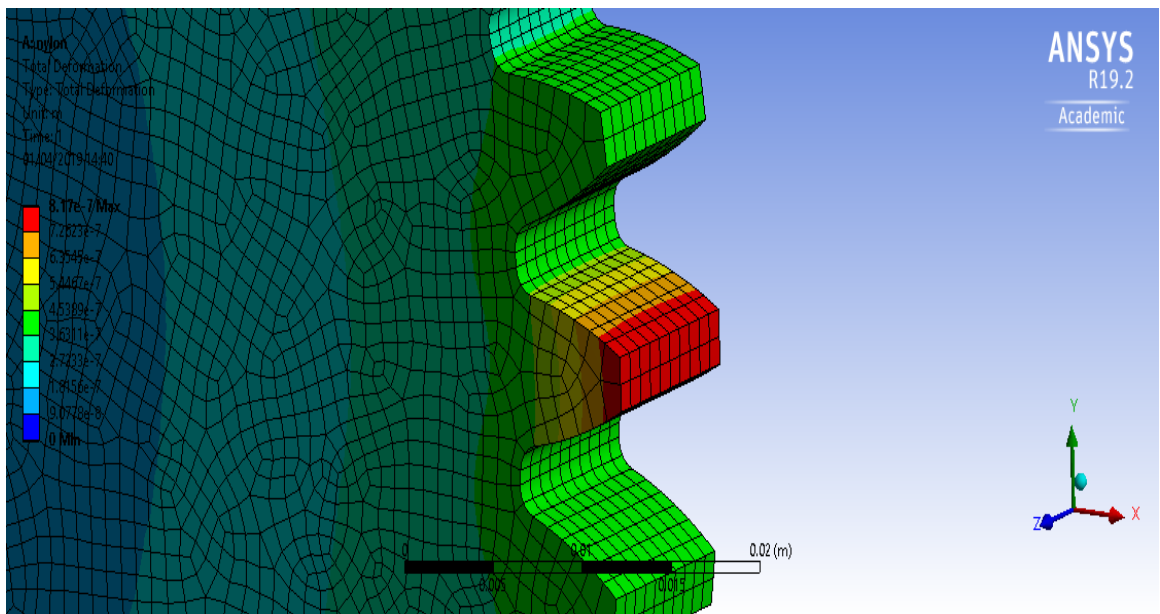
Fig 6.24 Maximum deformation representation of Nylon

54

6.2 CONTACT STRESS ANALYSIS OF SPUR GEAR

For Structural Steel

Normal Stresses
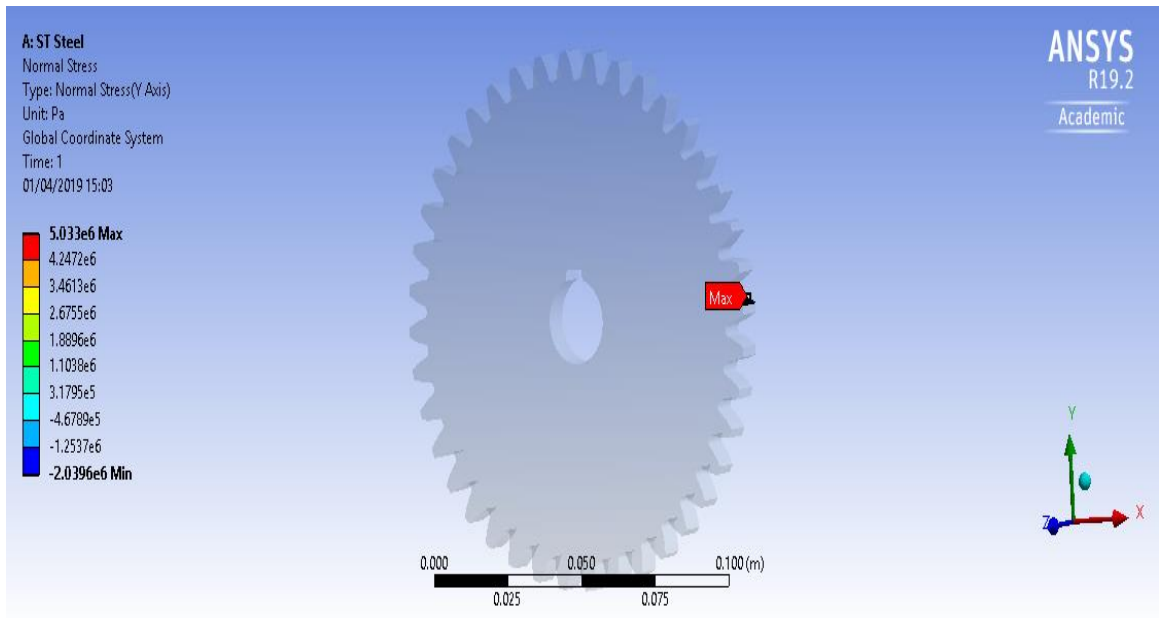
Fig 6.25 Contact stress on structural steel spur gear
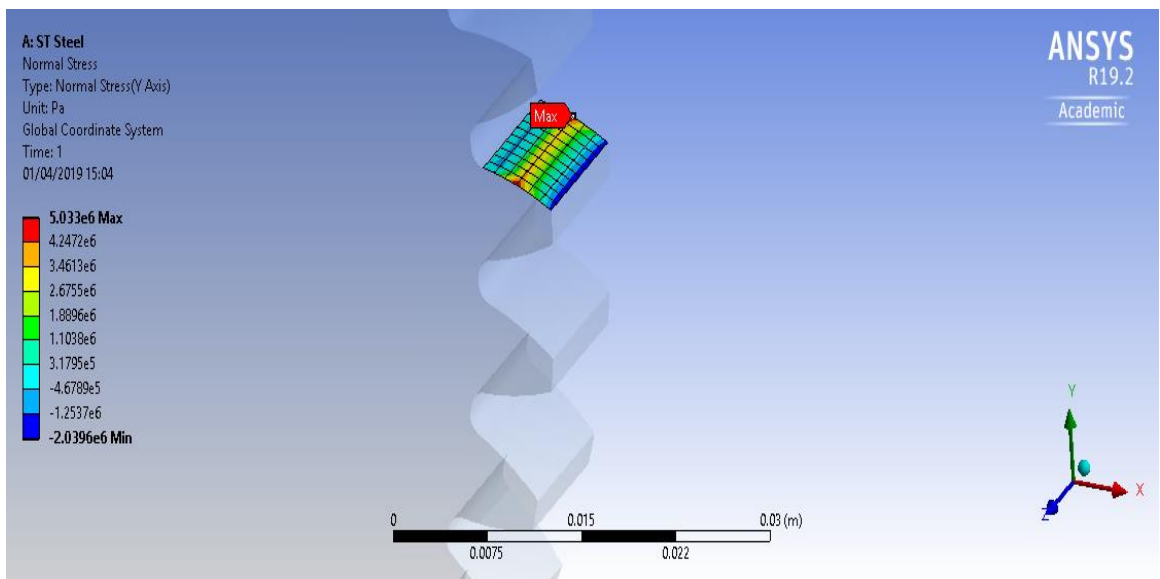
Fig 6.26 Contact stress on structural steel spur gear

Equivalent Stresses
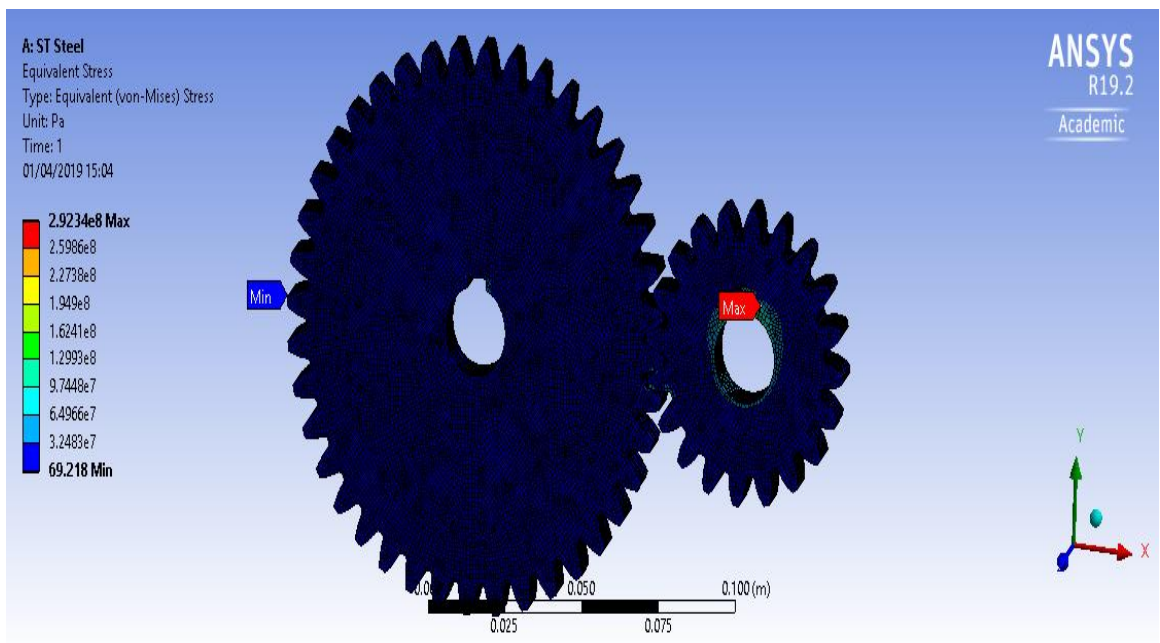
Fig 6.27 Equivalent stress on structural steel spur gear when in contact
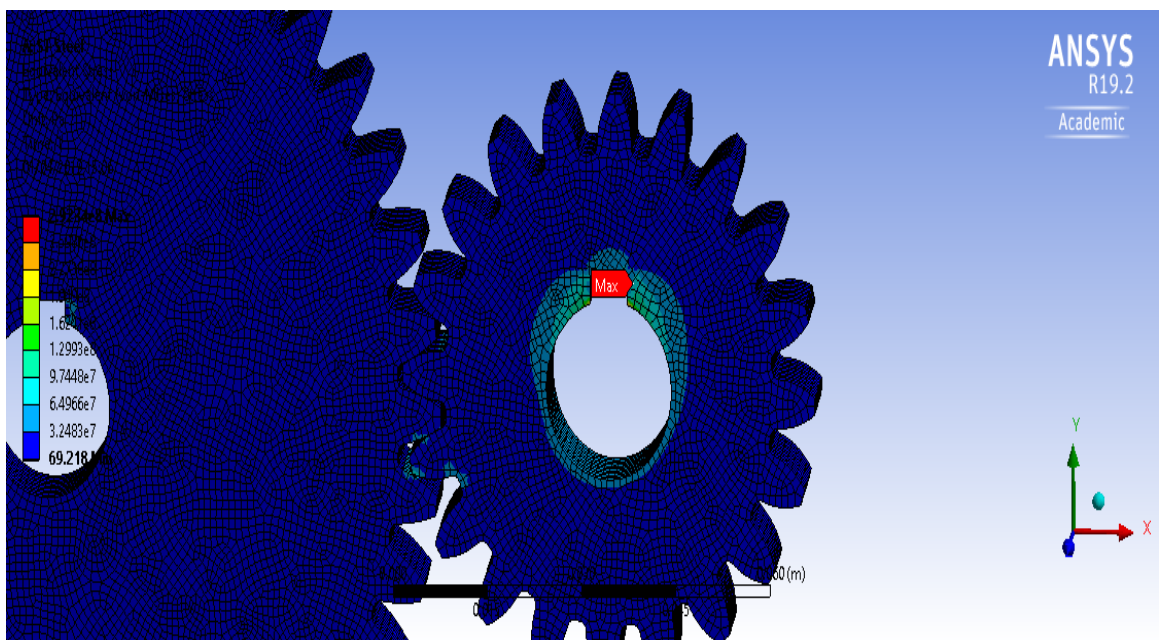
Fig 6.28 Representation of equivalent stress on structural steel spur gear

Normal Stresses 2
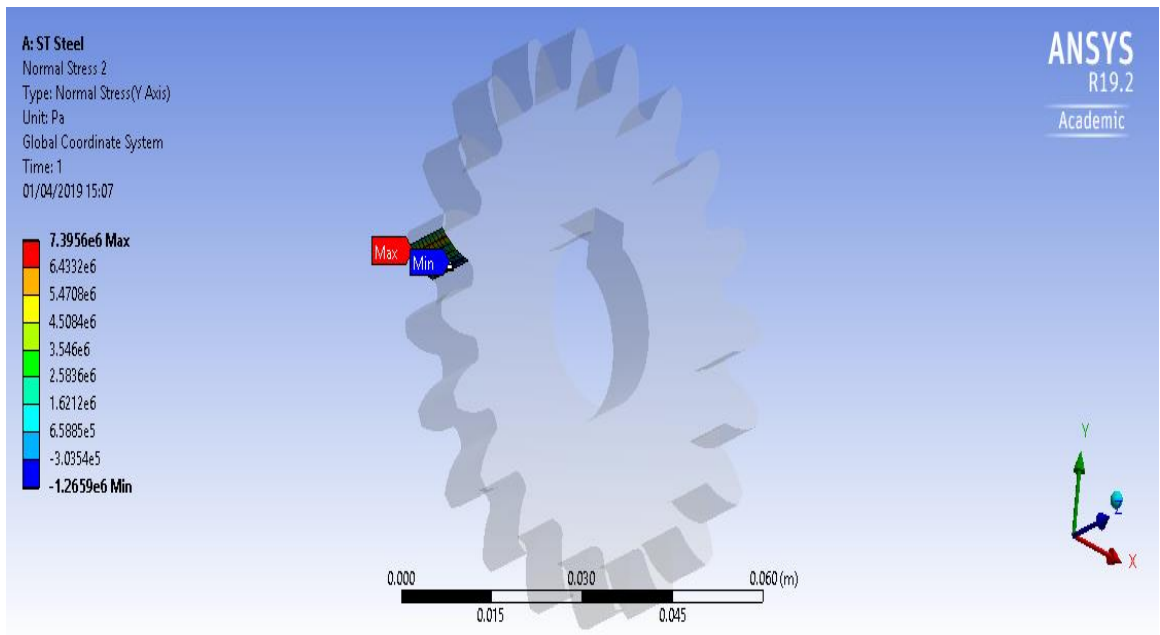
Fig 6.29 Normal stress due to contact stresses
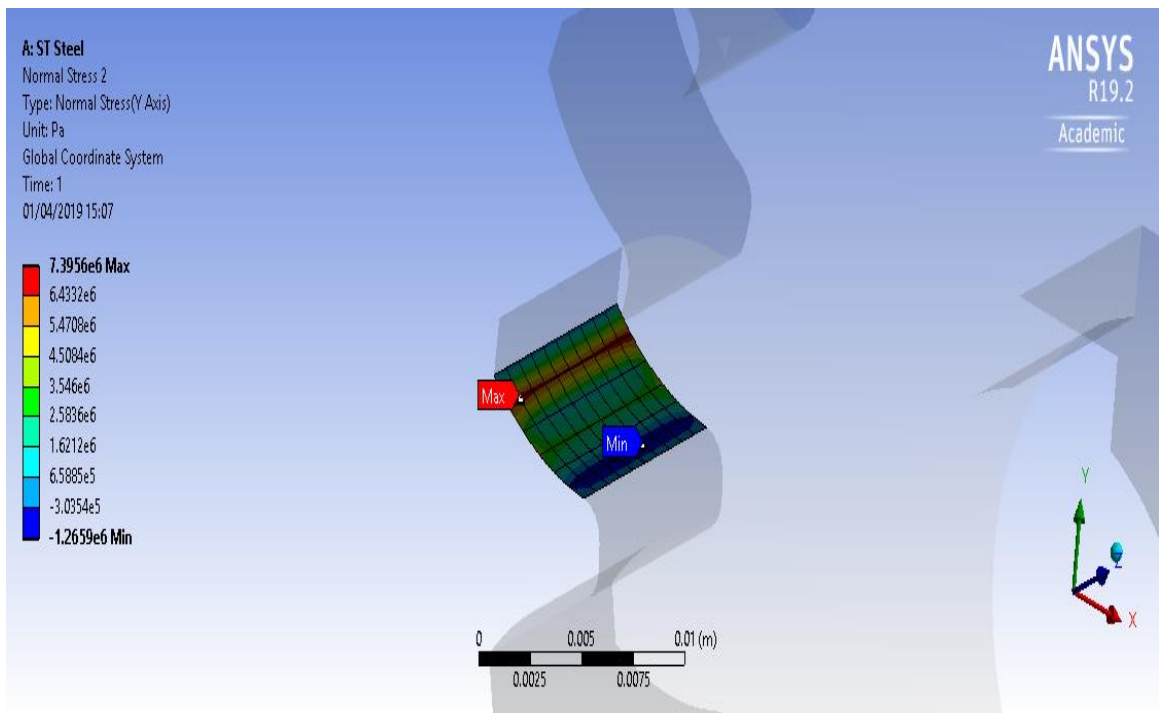
Fig 6.30 Representation of normal stress

Equivalent Stresses 2
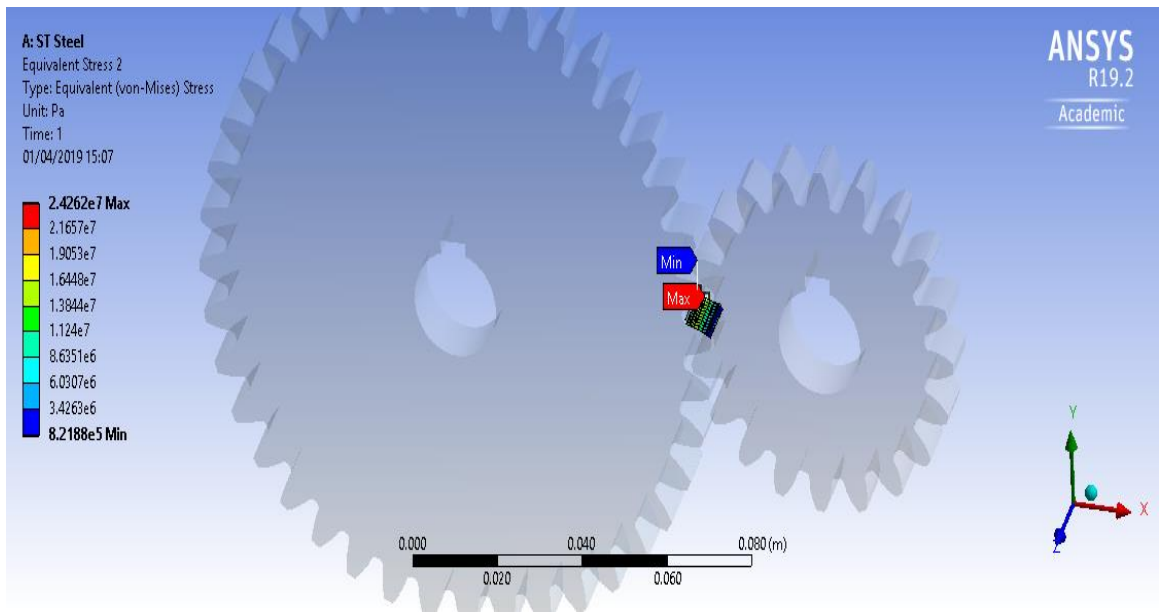
Fig 6.31 Equivalent stress on other part of contact surface
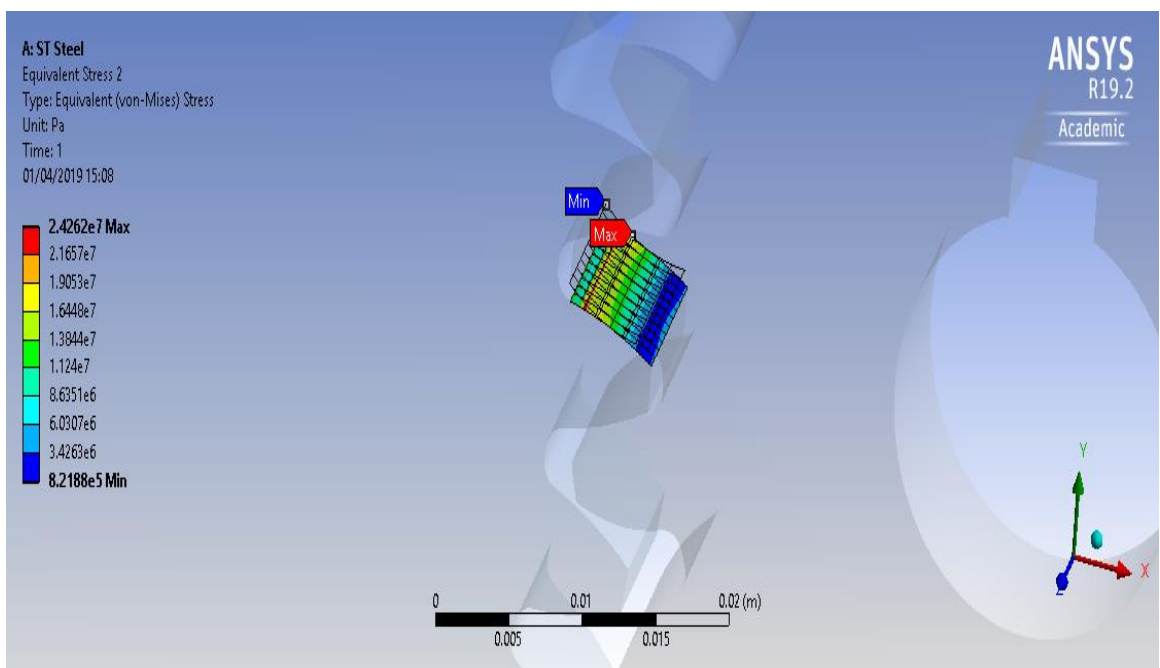
Fig 6.32 Representation of equivalent stress

6.3 STEADY STATE THERMAL ANALYSIS OF SPUR GEAR

6.3.1 For Structural Steel

Temperature
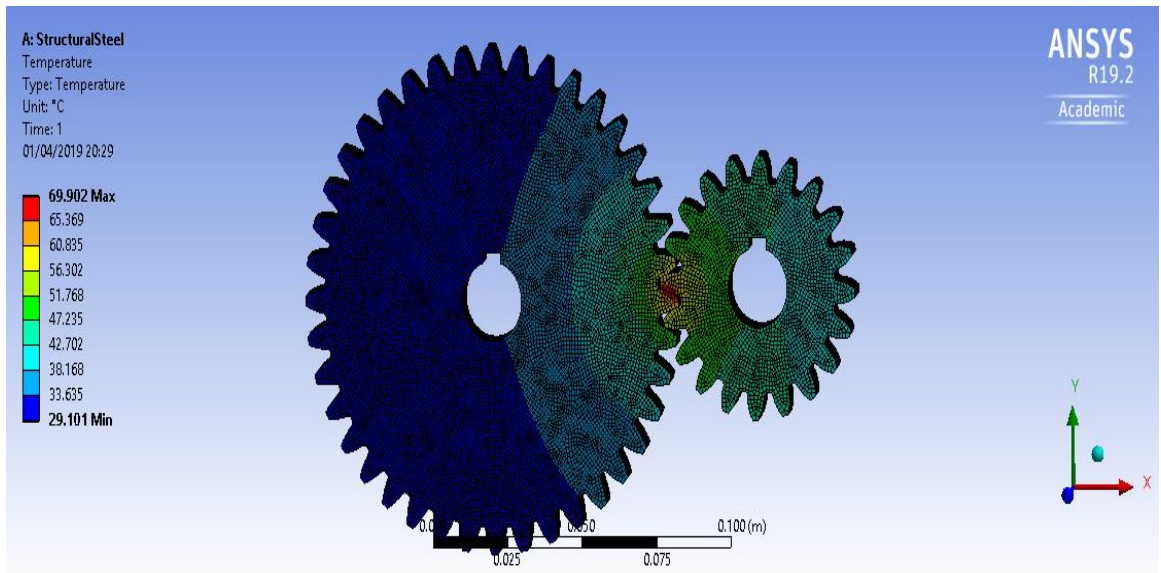
Fig 6.33 Maximum and minimum temperatures of structural steel
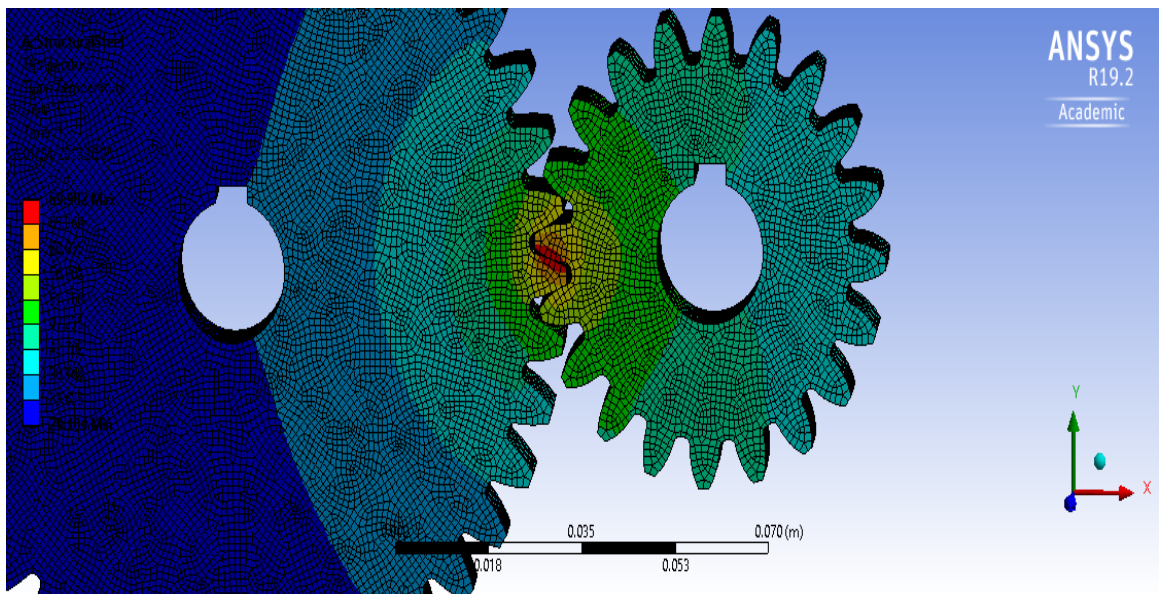
Fig 6.34 Representation of temperatures of structural steel

HeatFlux
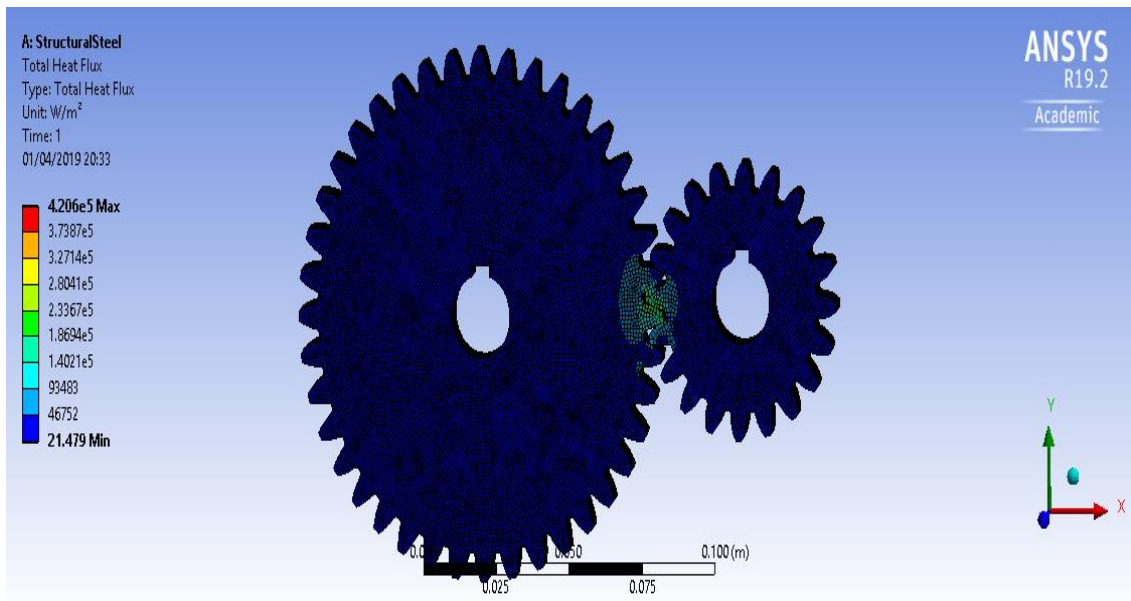
Fig 6.35 Heat flux on structural steel
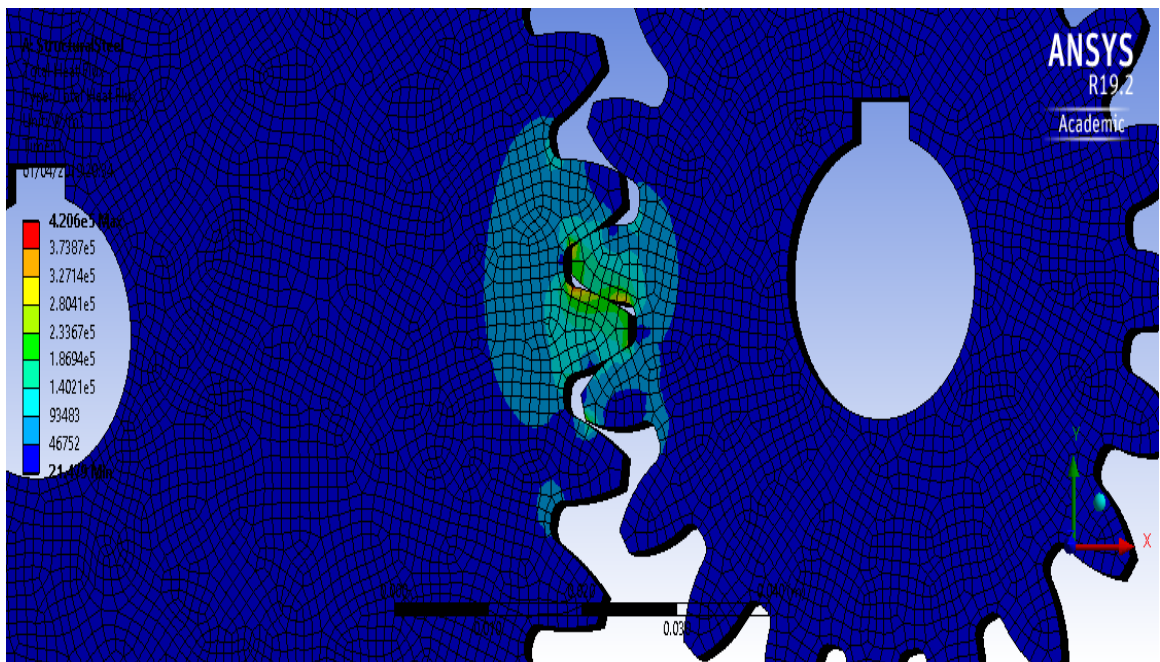
Fig 6.36 Representation of heat flux on structural steel

6.3.2 For Grey Cast Iron

Temperature
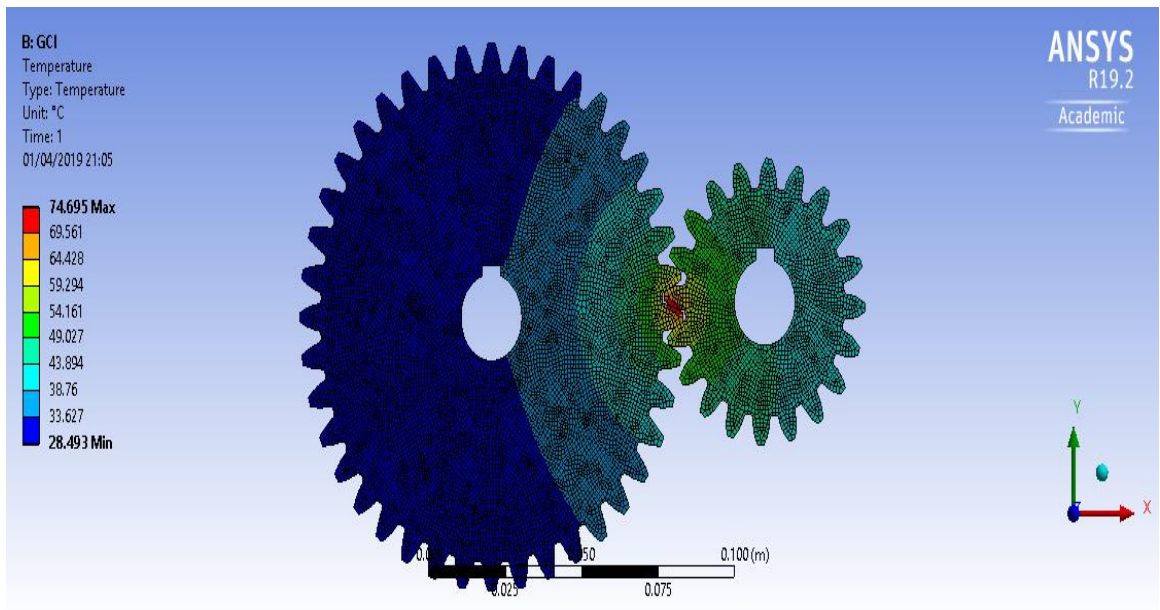
Fig 6.37 Maximum and minimum temperatures on grey cast iron
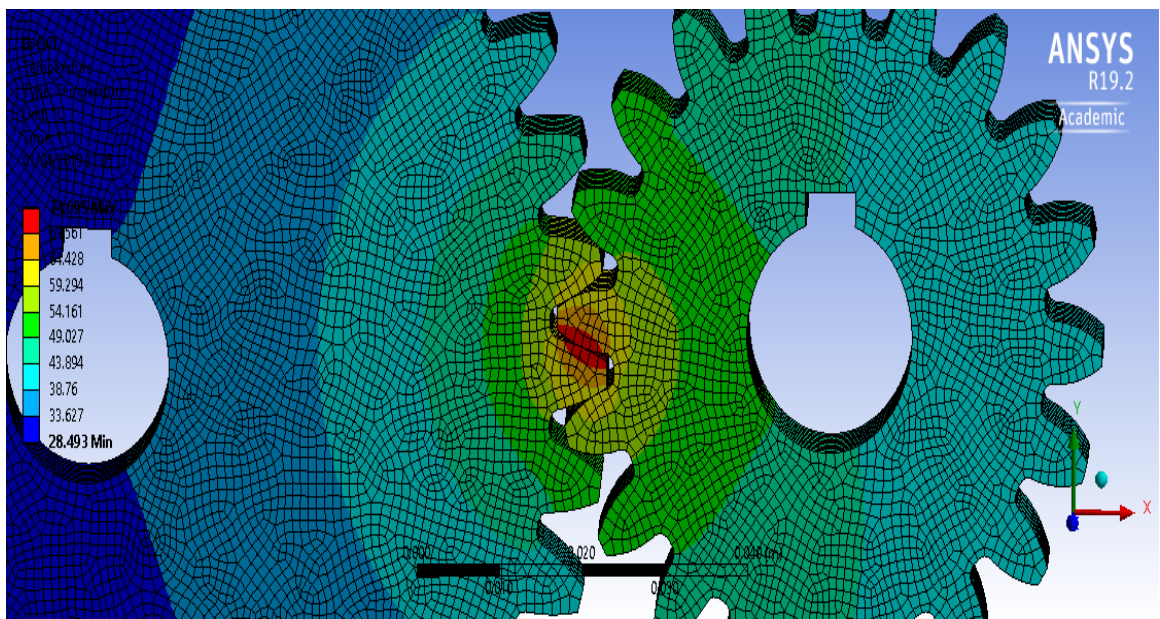
Fig 6.38 Representation of temperatures on grey cast iron

Heat Flux
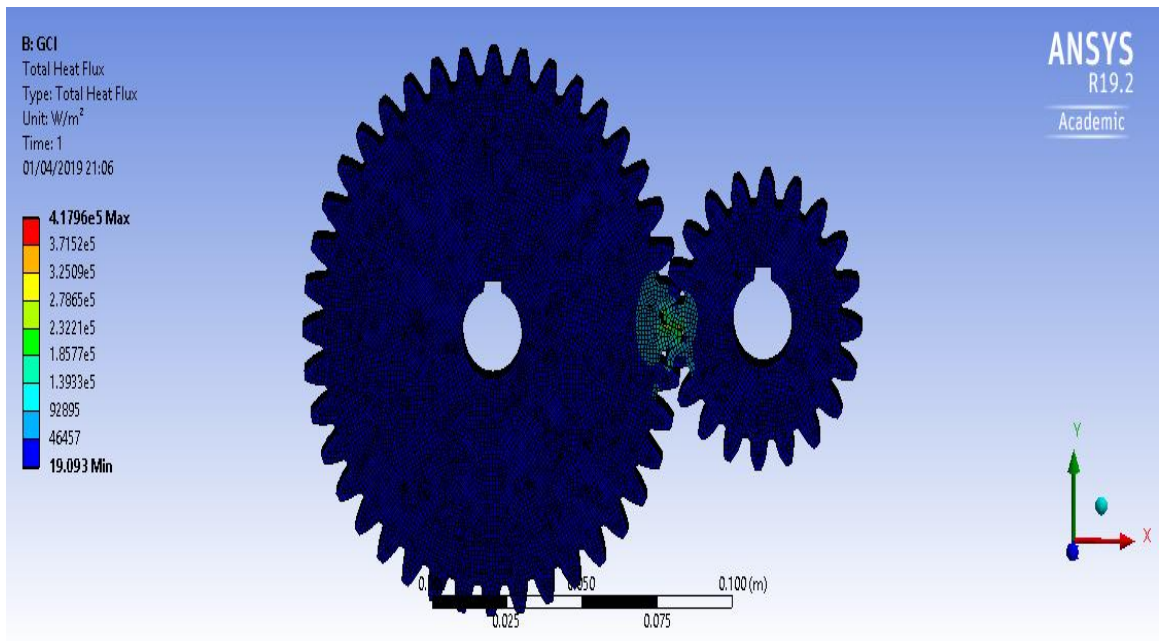
Fig 6.39 Heat flux on grey cast iron
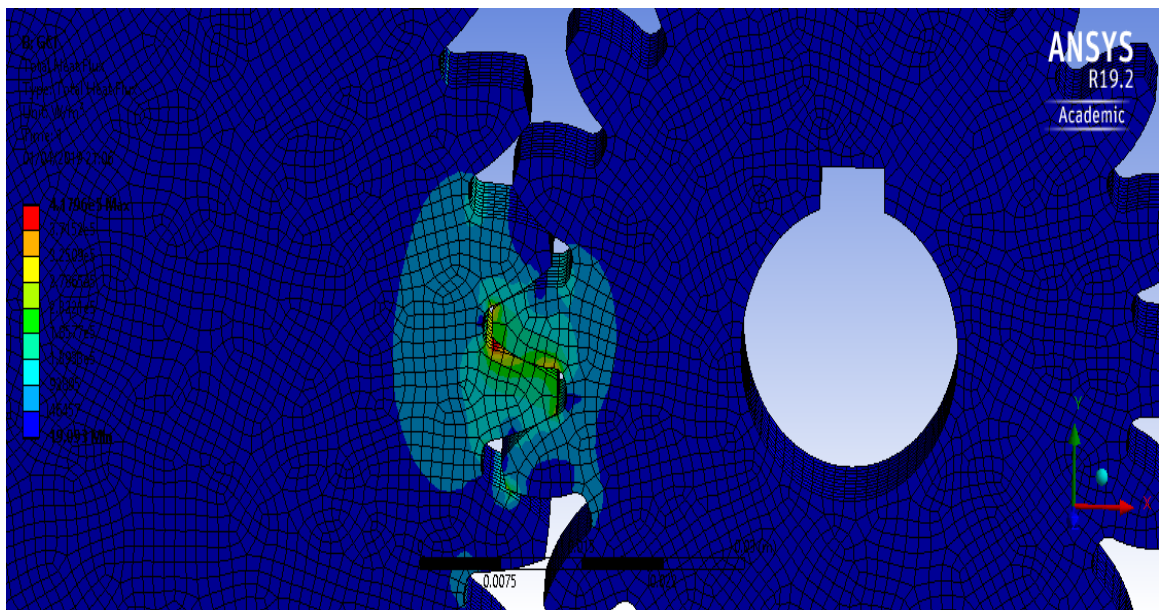
Fig 6.40 Representation of heat flux on grey cast iron

6.3.3 For CFRP

Temperature
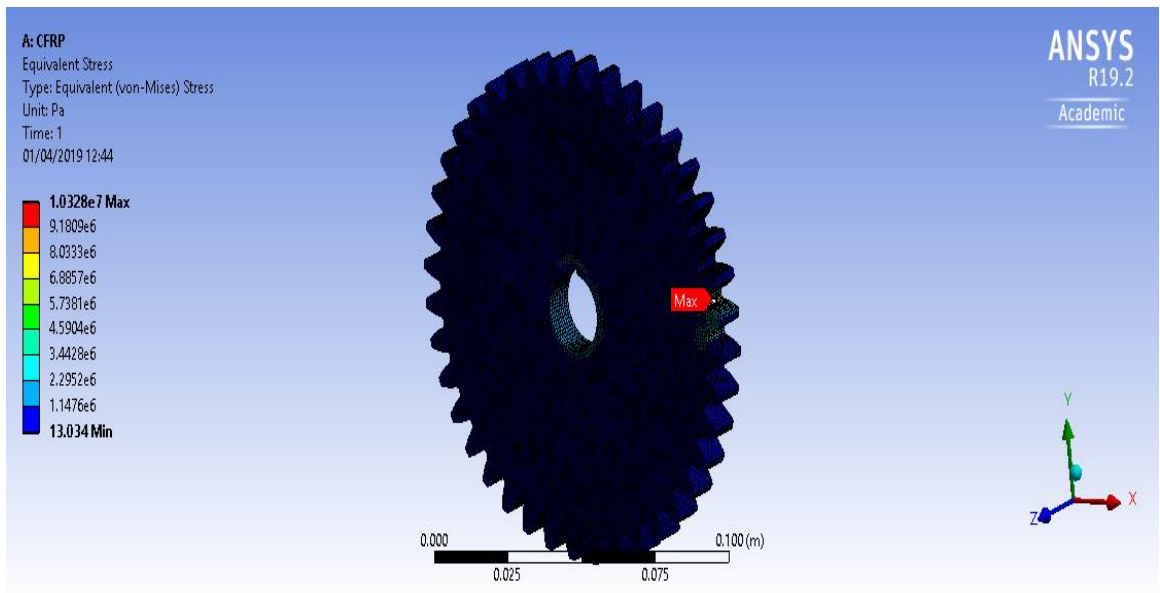
Fig 6.41 Maximum and minimum temperatures on CFRP
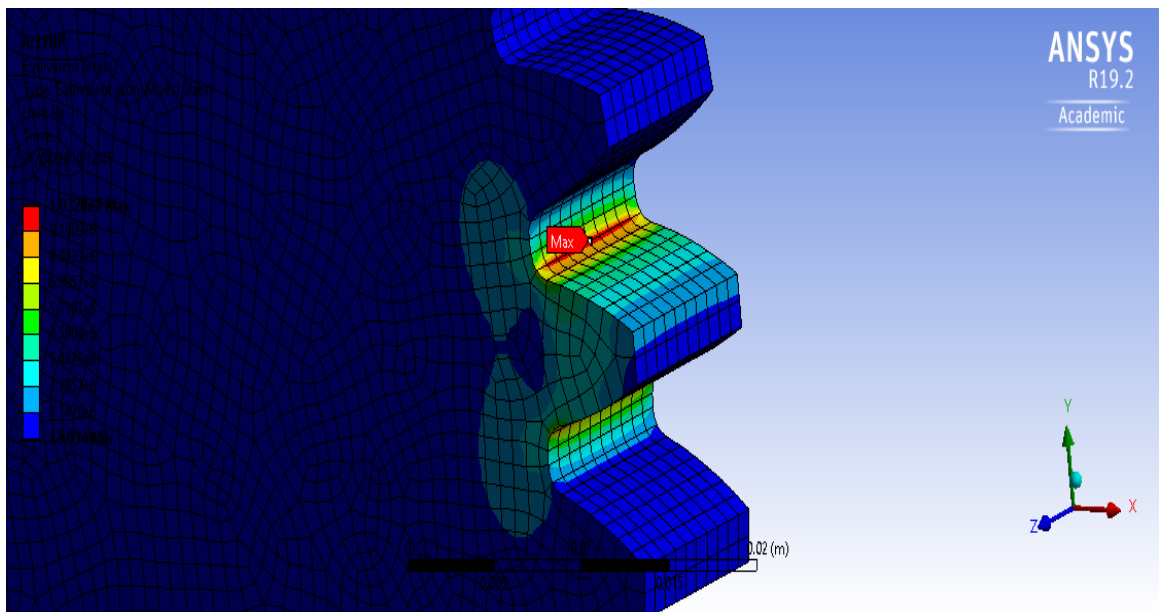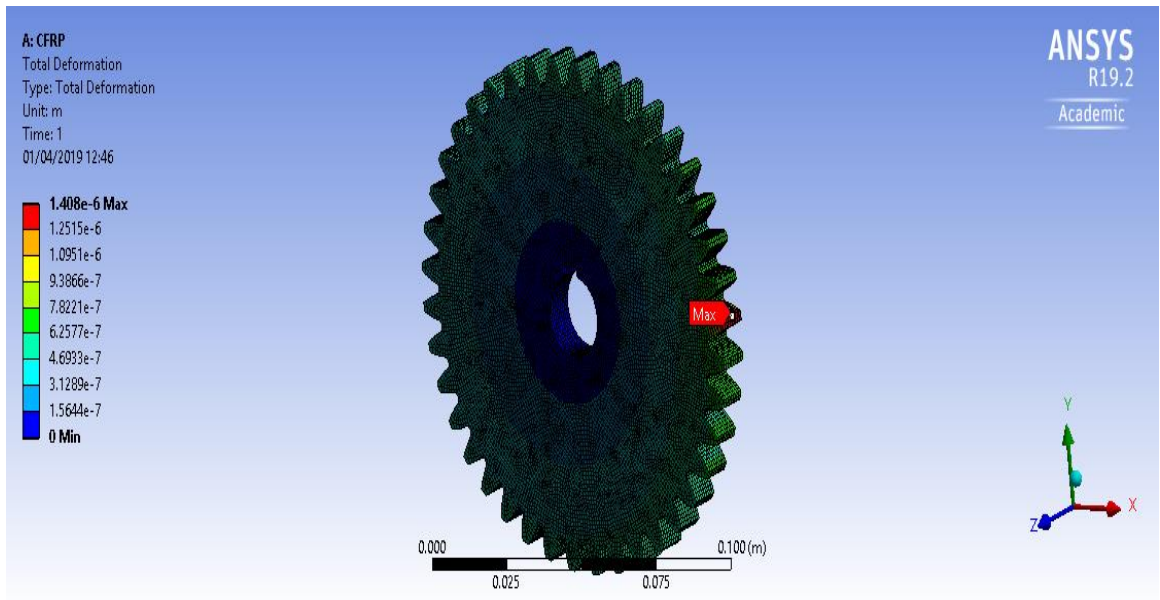

Fig 6.42 Representation of temperatures onCFRP
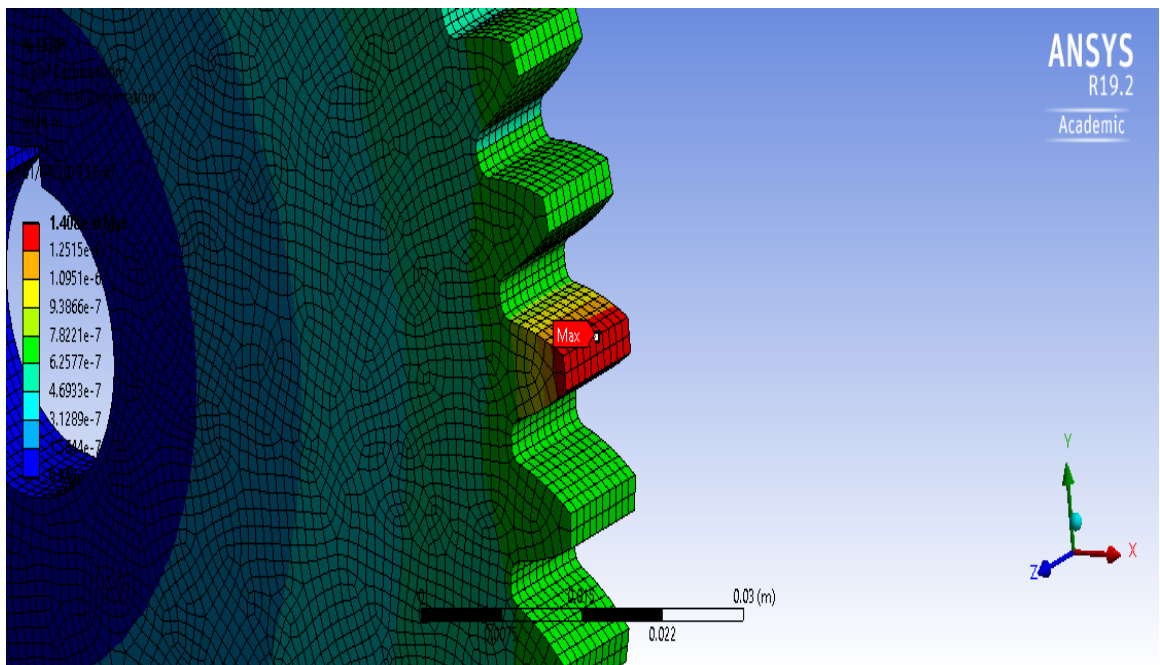
Heat Flux

Fig 6.43 Heat flux on CFRP

Fig 6.44 Representation of heat flux on CFRP

64

6.3.4 For Al SiC 15%
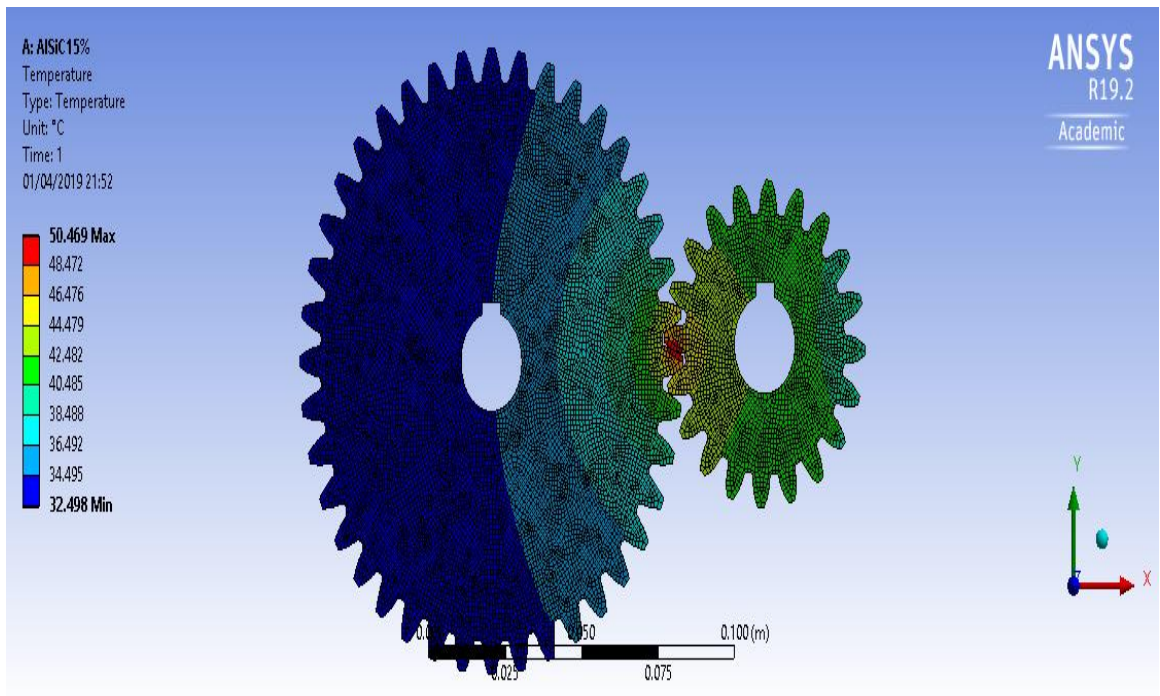
Temperature

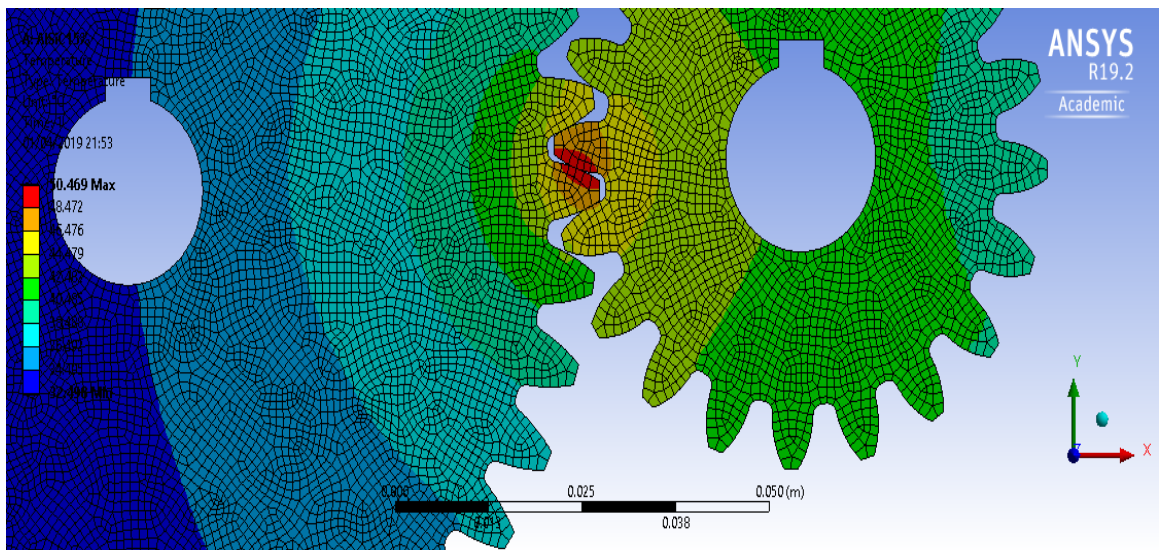**Fig 6.45** Maximum and minimum temperatures on Al SiC 15%

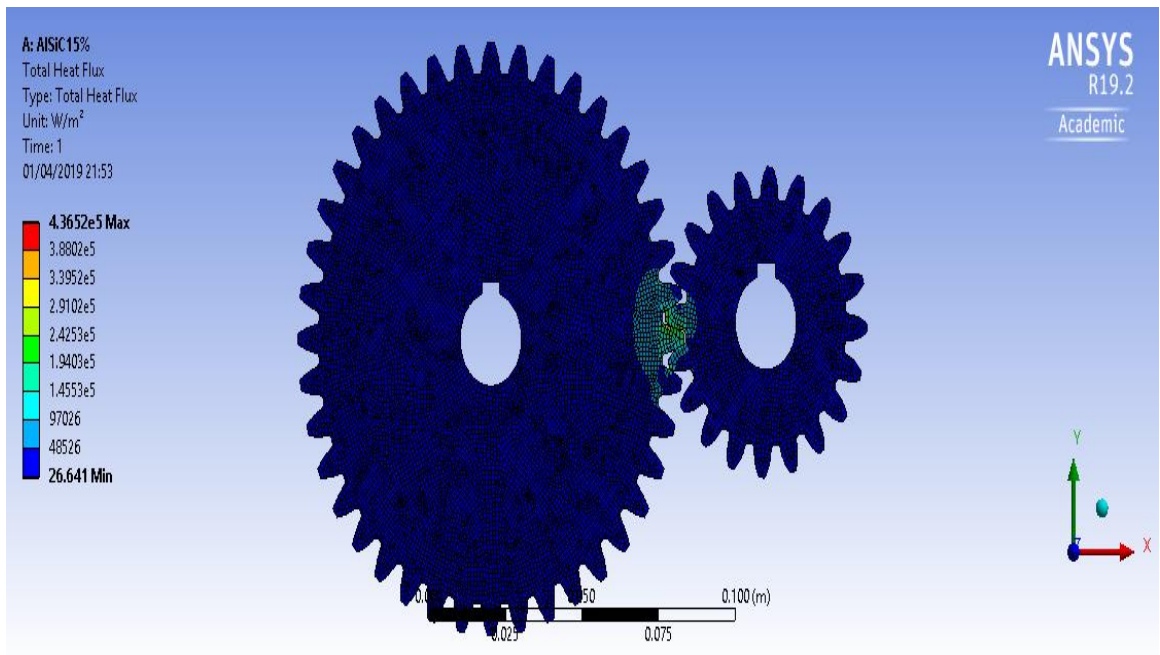**Fig 6.46** Representation of temperatures on Al SiC 15%

Heat Flux

Fig 6.47 Heat flux on Al SiC 15%
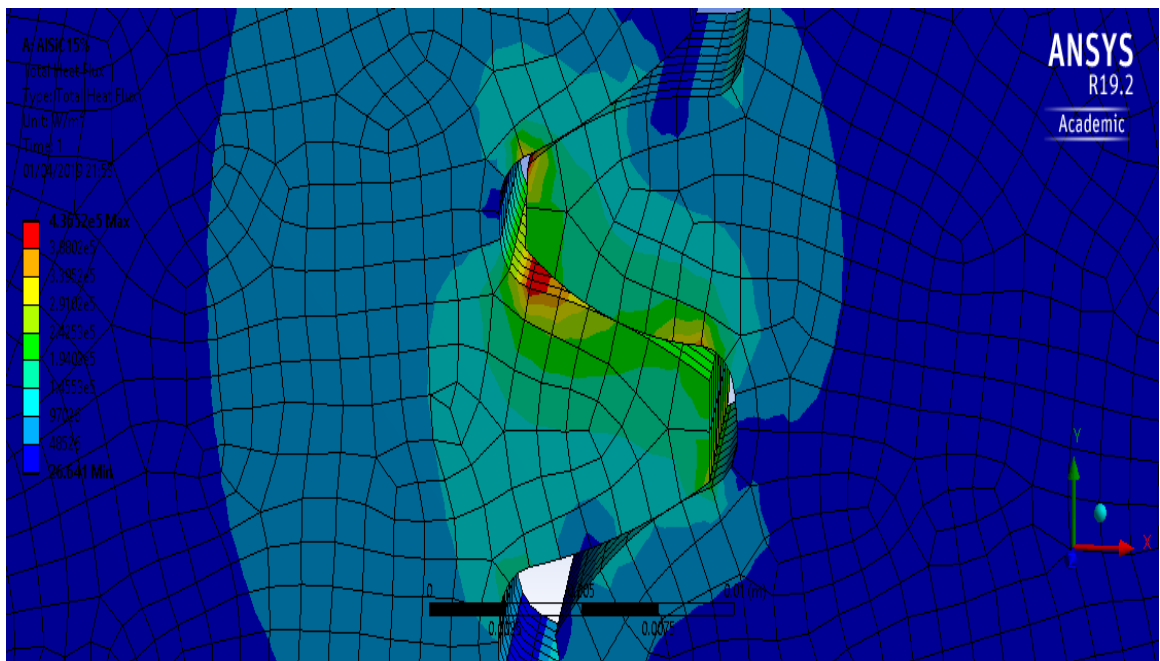

Fig 6.48 Representation of heat flux on Al SiC 15%

6.3.5 For Al SiC 16%, CNT 1%
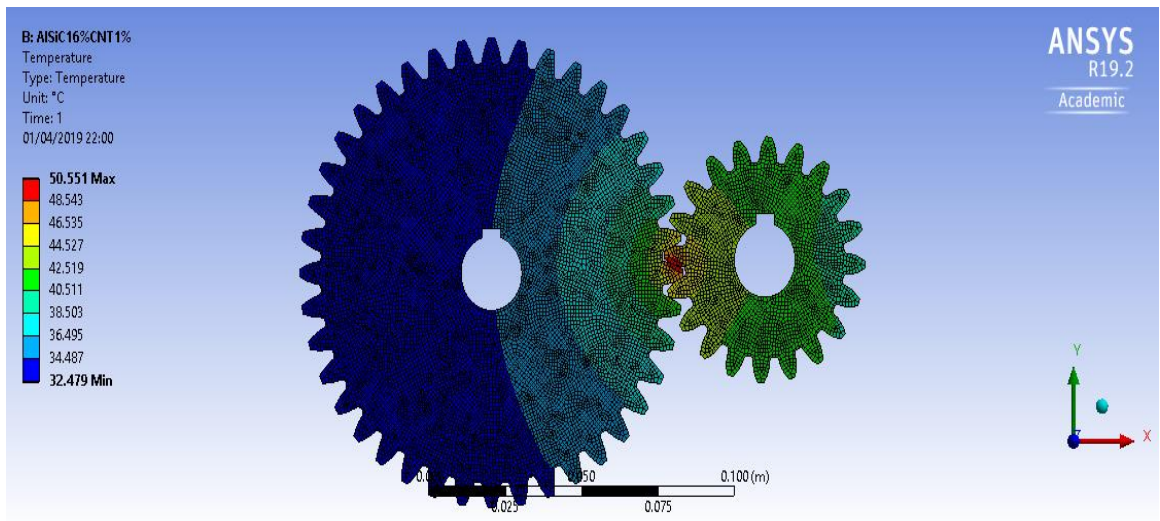
Temperature

Fig 6.49 Maximum and minimum temperatures on Al SiC 16% CNT 1%
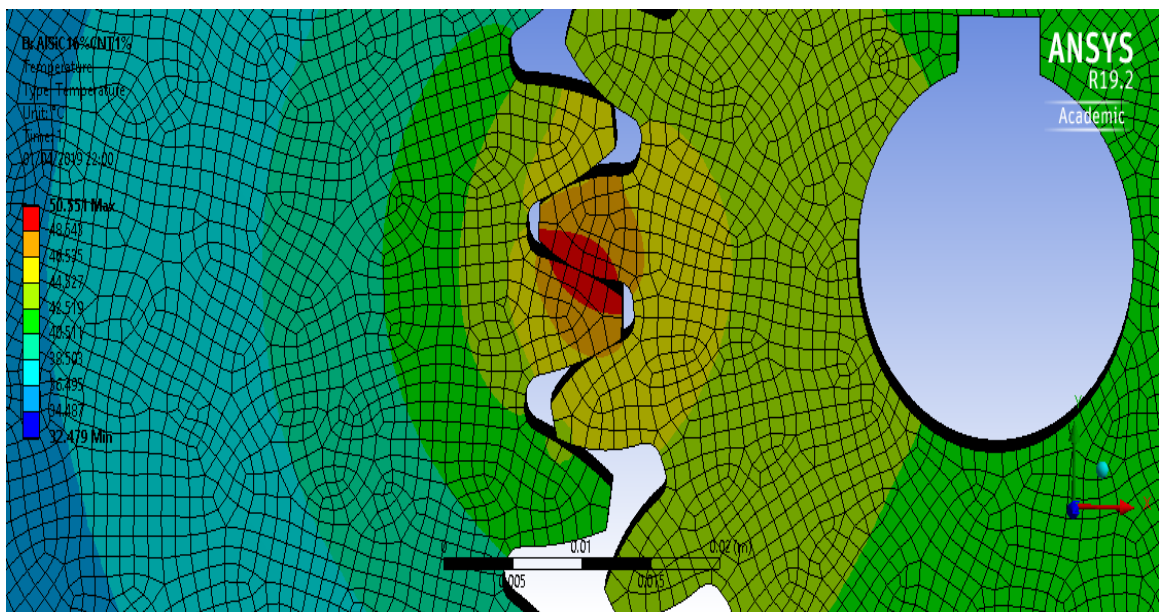

Fig 6.50 Representation of temperatures on Al SiC 16% CNT 1%
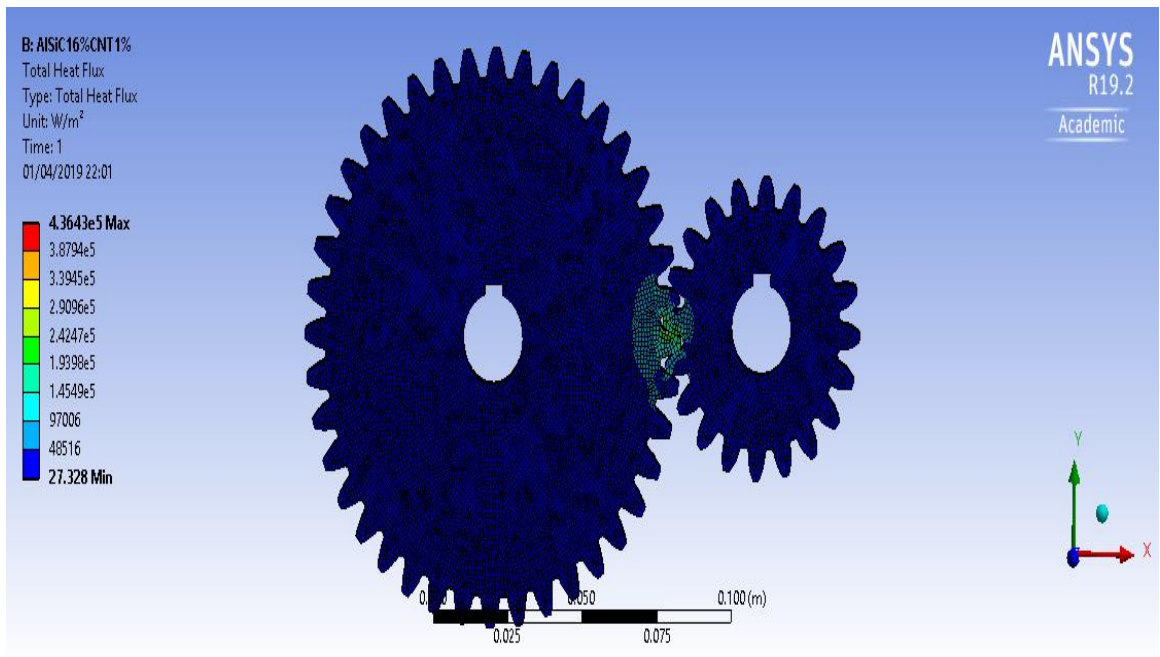
Heat Flux

Fig6.51 Heat flux on Al SiC 16% CNT 1%
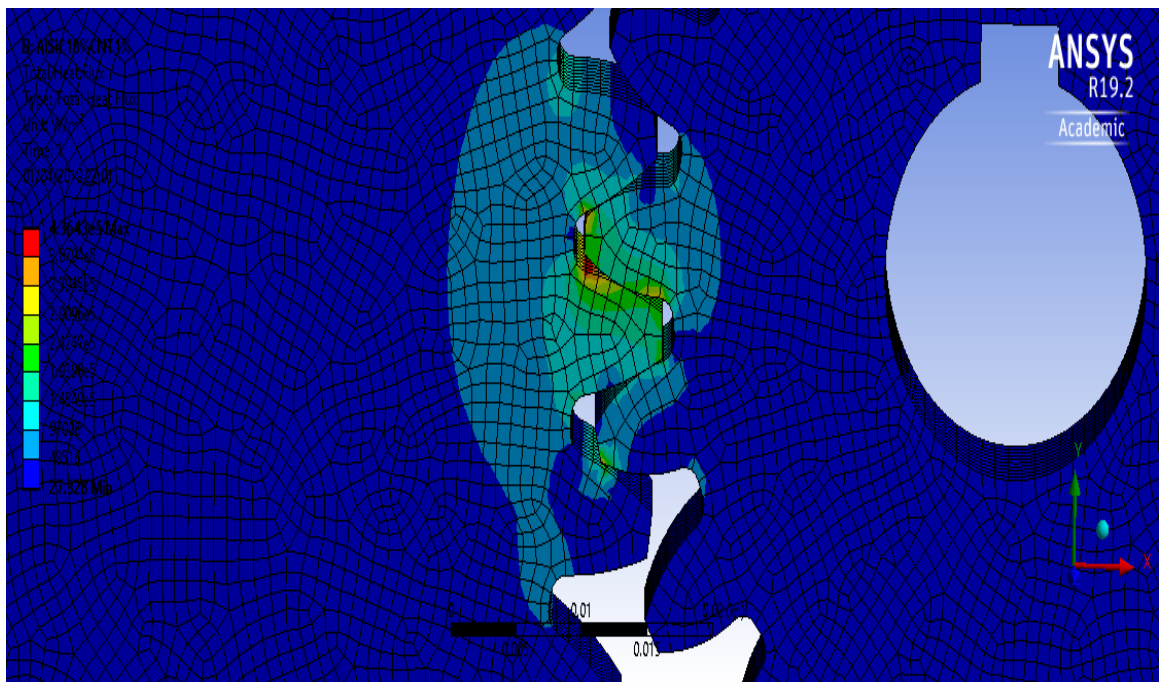
Fig 6.52 Representation of heat flux on Al SiC 16% CNT 1%

6.3.6 For Nylon

Temperature
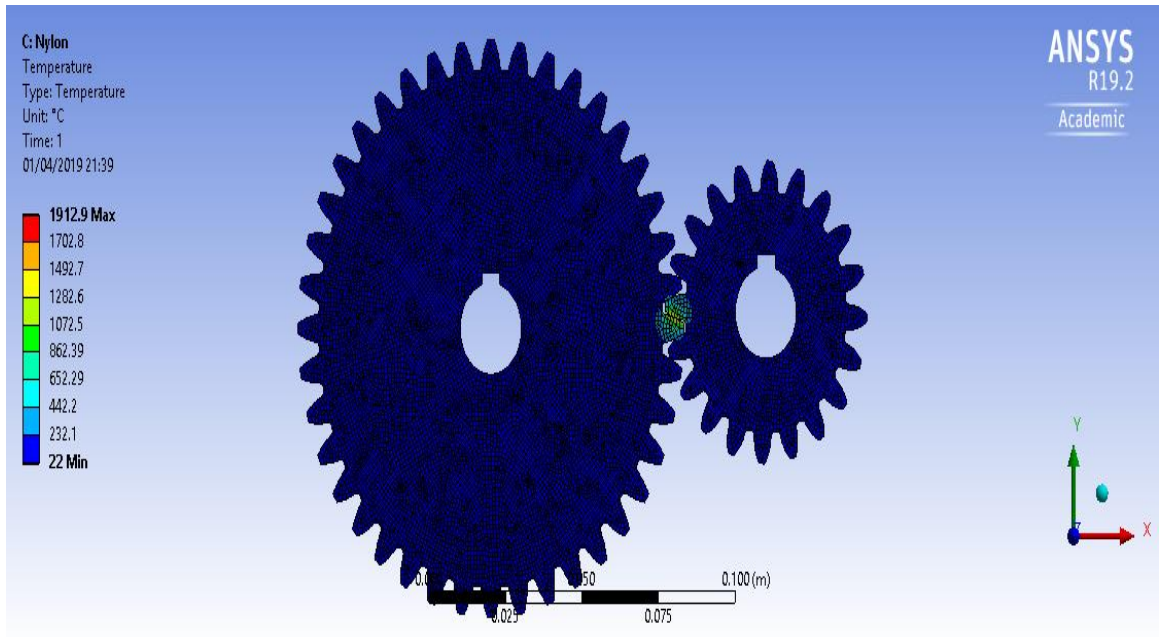
Fig 6.53 Maximum and minimum temperatures on Nylon
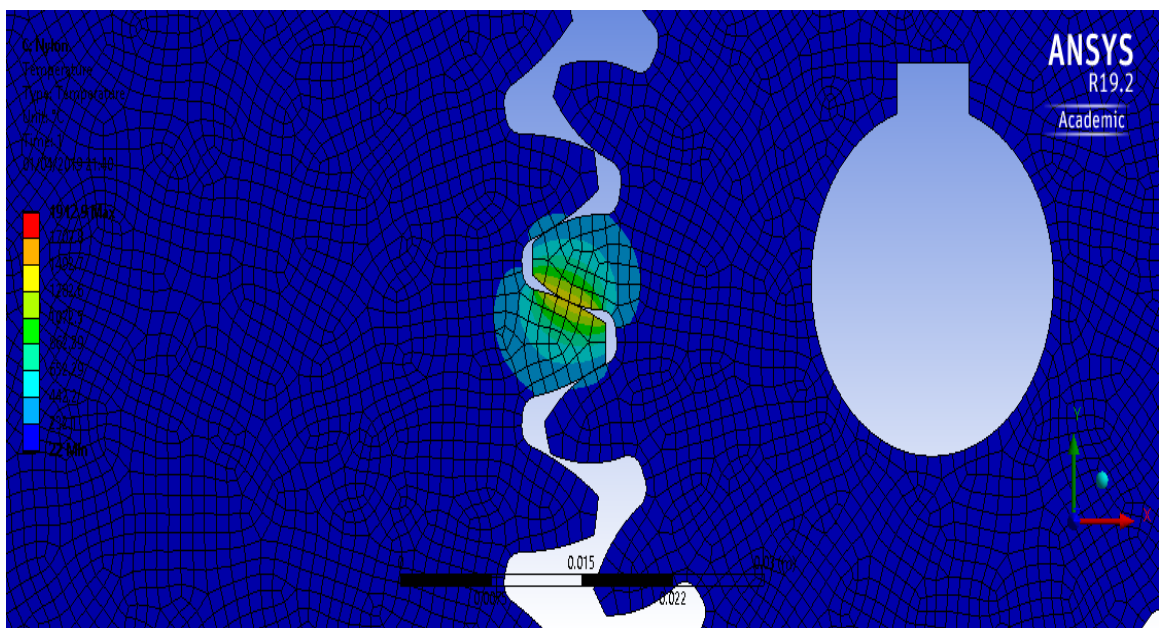
Fig 6.54 Representation of temperatures on Nylon

Heat Flux
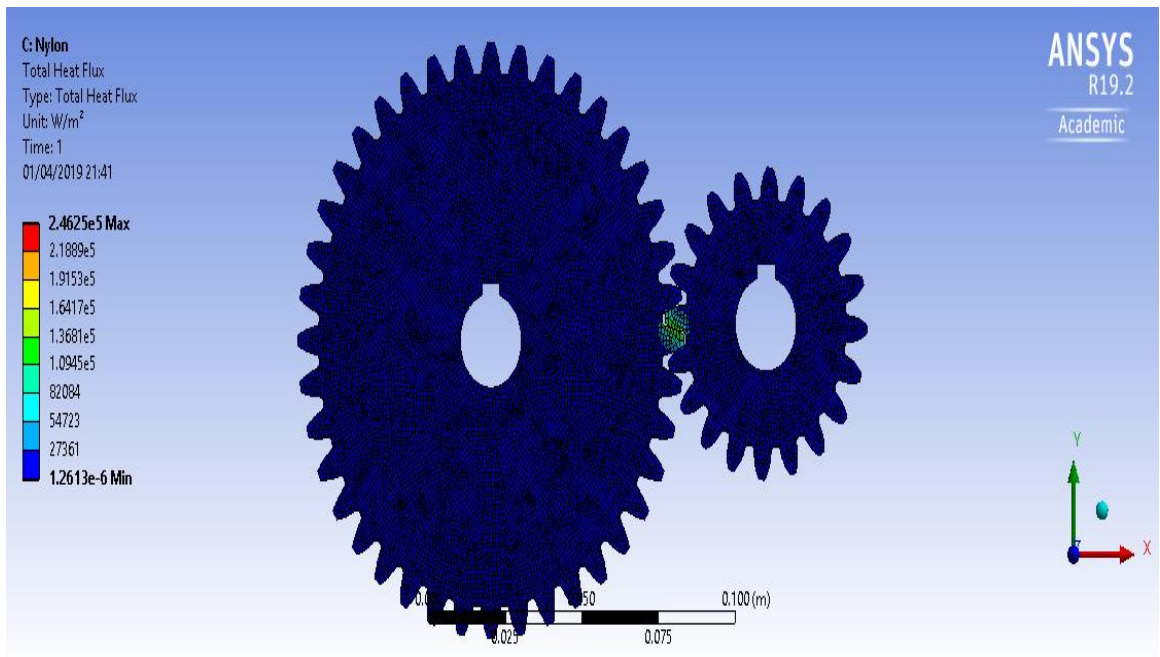
Fig 6.55 Heat flux on Nylon
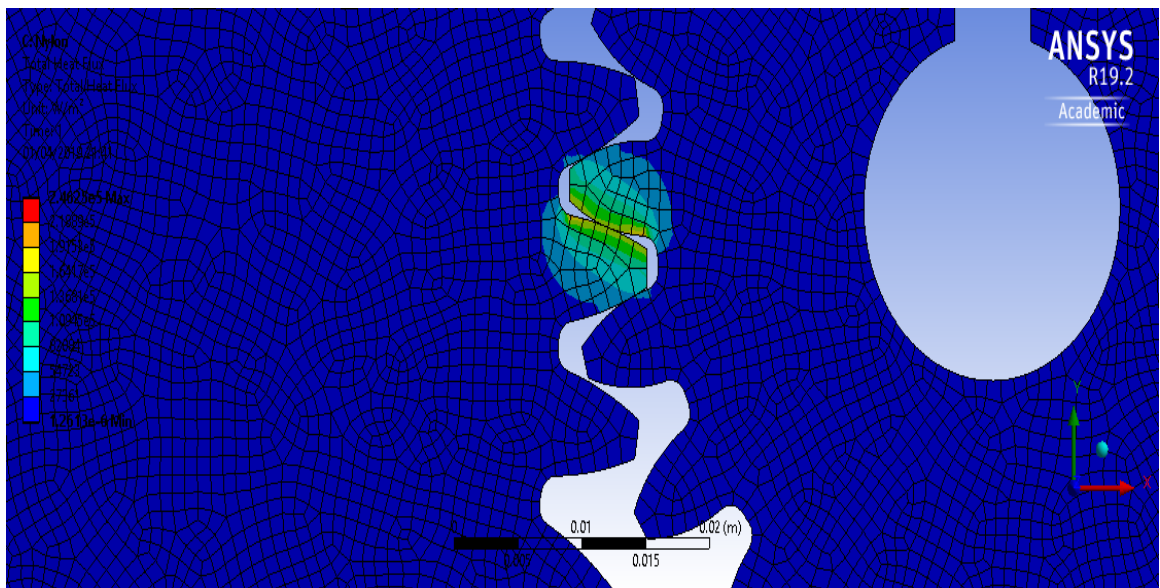
Fig 6.56 Representation of heat flux on Nylon

The contact stress analysis is done only for structural steel because the other materials are failing for the given load conditions and even the structural steel is modified by eliminated by removing fixed support and displacement

6.4 Comparison among different materials used for the gear

70

6.4.1 Equivalent stresses in static structural conditions

TABLE 6.1 Comparison of equivalent stresses for STEEL and GCI

|	STEEL	GCI
Force Y Component (N)	Equivalent Stress Maximum (MPa)	Equivalent Stress Maximum (MPa)
100	10.18	10.1685714
300	30.55	30.50571403
500	50.91	50.84285528
700	71.28	71.1799983
900	91.65	91.51714167
1100	112.02	111.8542838
1300	132.38	132.1914266
1500	152.75	152.5285665

TABLE 6.2 Comparison of equivalent stresses for CFRP and Al SiC 15%

	CFRP	**Al SiC 15%**
Force Y Component (N)	Equivalent Stress Maximum (MPa)	Equivalent Stress Maximum (MPa)
100	10.32848416	10.18372638
300	30.98545423	30.55117933
500	51.64242055	50.91863293
700	72.29939321	71.28608336
900	92.95635608	91.65353844
1100	113.6133254	112.0209905
1300	134.2702982	132.3884421
1500	154.9272677	152.7558976

TABLE 6.3 Comparison of equivalent stresses of Al SiC 16% CNT 1% and Nylon

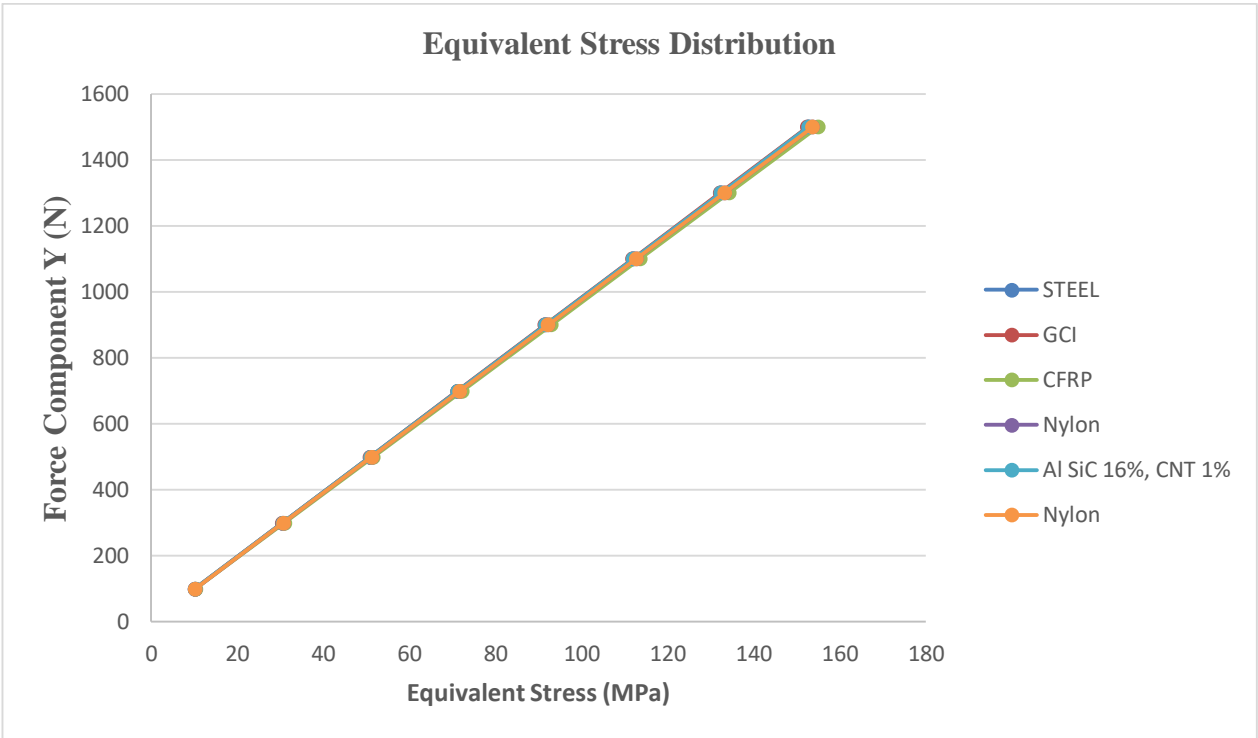|	Al SiC 16%, CNT 1%	Nylon
Force Y Component (N)	Equivalent Stress Maximum (MPa)	Equivalent Stress Maximum (MPa)
100	10.18372635	10.24854944
300	30.55117944	30.74564985
500	50.91863314	51.24274844
700	71.28608344	71.73984721
900	91.6535386	92.2369462
1100	112.0209913	112.7340446
1300	132.3884425	133.2311406
1500	152.7558912	153.728243

Fig 6.57 Graphical representation of equivalent stress

6.4.2 Total deformation in static structural conditions

TABLE 6.4 Comparison of deformation of STEEL and GCI

Force Y Component (**N**)	Steel Total Deformation Maximum (**mm**)	GCI Total Deformation Maximum (**mm**)
100	0.000905803	0.00163689
300	0.002717408	0.004910669
500	0.004529013	0.008184448
700	0.006340618	0.011458228
900	0.008152223	0.014732008
1100	0.009963828	0.018005786
1300	0.011775433	0.021279567
1500	0.013587038	0.024553347

TABLE 6.5 Comparison of deformation of CFRP and Al SiC 15%

Force Y Component (**N**)	CFRP Total Deformation Maximum (**mm**)	Al SiC 15% Total Deformation Maximum (**mm**)
100	0.001407984	0.001207737
300	0.004223952	0.00362321
500	0.007039921	0.006038683
700	0.009855889	0.008454157
900	0.012671857	0.01086963
1100	0.015487826	0.013285104
1300	0.018303794	0.015700578
1500	0.021119762	0.018116052

TABLE 6.6 Comparison of deformation of AlSiC16%CNT1% and Nylon

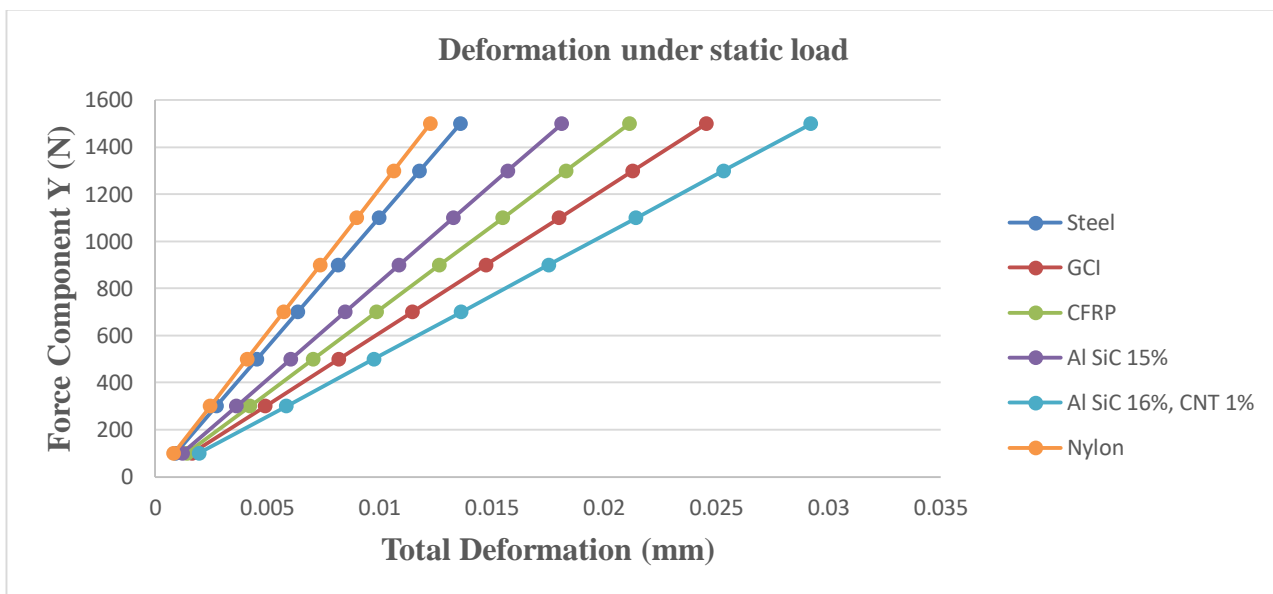| Force Y Component (**N**)	Al Sic 16%, CNT 1% Total Deformation Maximum (**mm**)	Nylon Total Deformation Maximum (**mm**)
100	0.001947962	0.000817005
300	0.005843887	0.002451014
500	0.009739813	0.004085023
700	0.013635737	0.005719033
900	0.017531663	0.007353042
1100	0.021427586	0.008987051
1300	0.025323512	0.010621061
1500	0.029219437	0.01225507

Fig 6.58 Graphical representation of deformation

6.5 Comparison among different materials in steady state thermal analysis

6.5.1 Temperature distribution

TABLE 6.7 Comparison of Temperatures

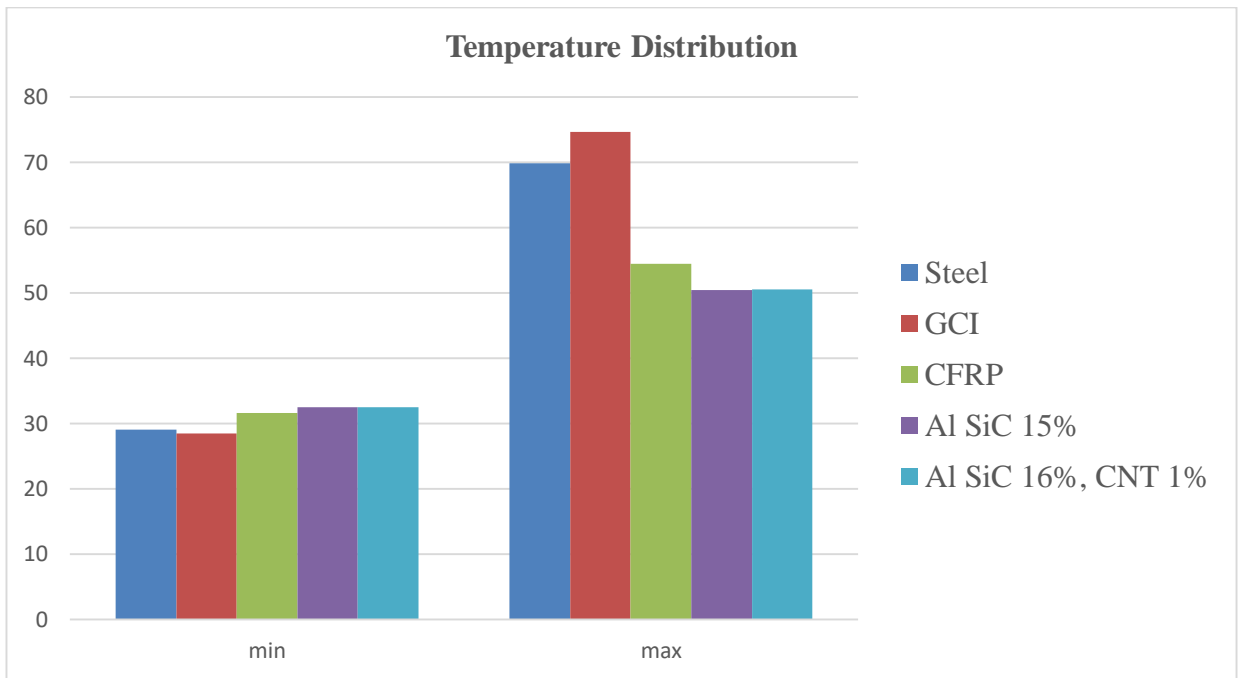| Materials	Temperature(°C) Minimum	Temperature(°C) Maximum
Steel	29.101	69.902
GCI	28.493	74.695
CFRP	31.633	54.513
Al SiC 15%	32.498	50.469
Al SiC 16%, CNT 1%	32.479	50.551
Nylon	22	1912.9

Fig 6.59 Graphical representation of maximum and minimum temperatures

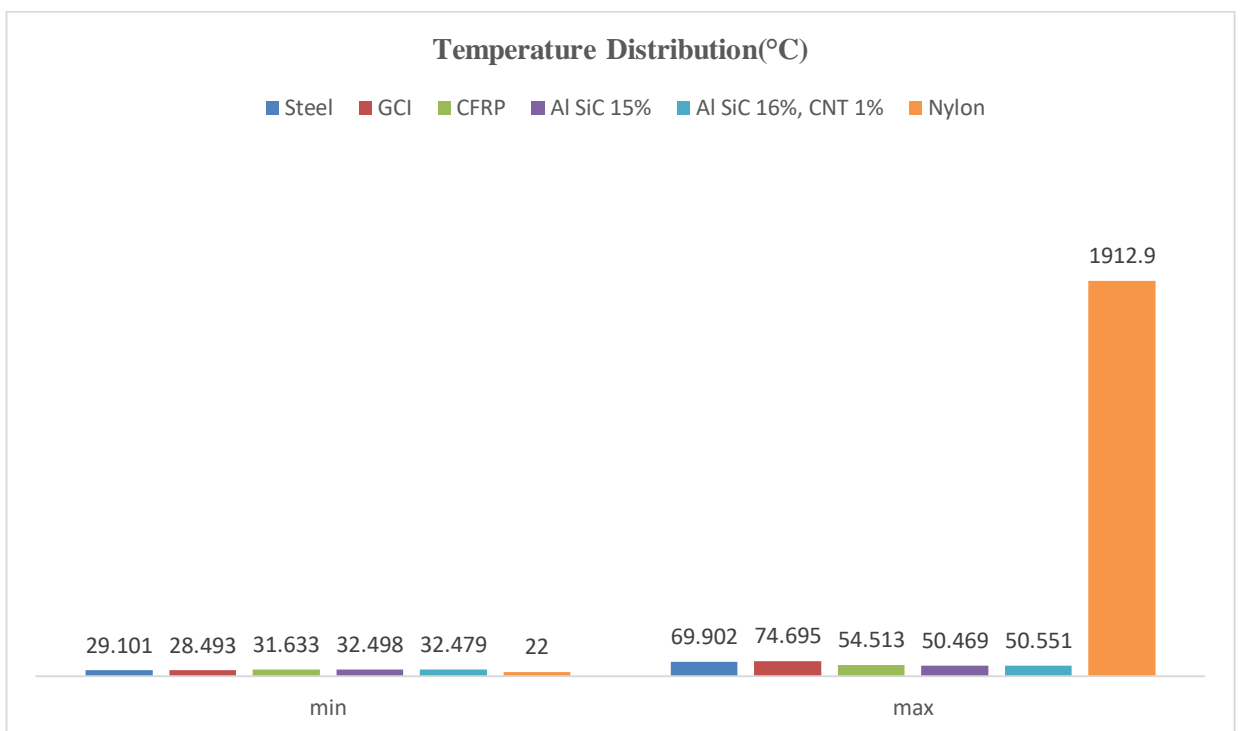**Fig 6.60** Graphical representation of maximum and minimum heat flux

6.52 Heat Flux distribution

TABLE 6.8 Comparison of heat flux

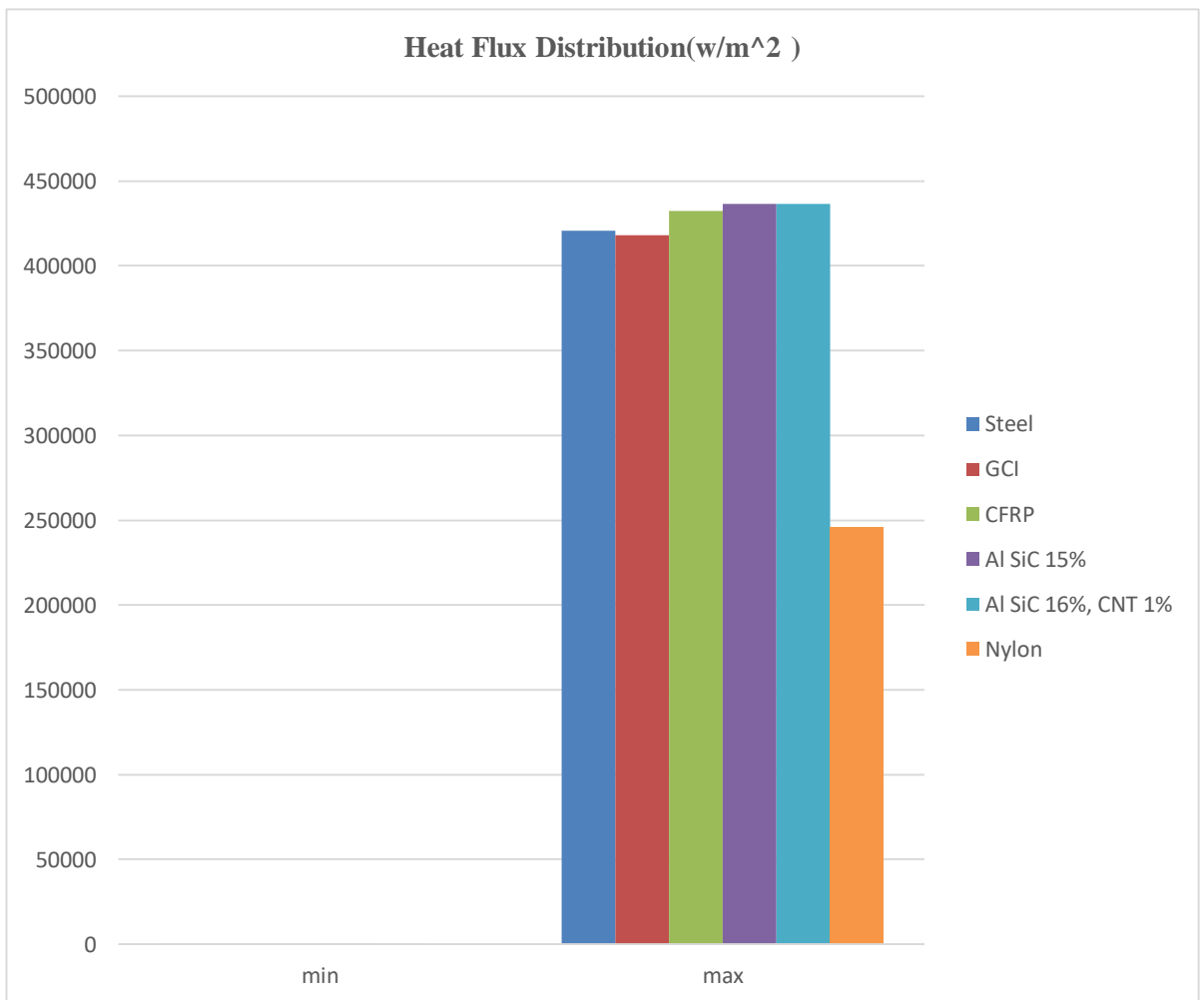| Materials	Heat Flux (w/m^2) Minimum	Heat Flux (w/m^2) Maximum
Steel	21.479	4.21E+05
GCI	19.093	4.18E+05
CFRP	35.545	4.32E+05
Al SiC 15%	26.641	4.37E+05
Al SiC 16%, CNT 1%	27.328	4.36E+05
Nylon	1.26E-06	2.46E+05

Fig 6.61 Graphical representation of maximum and minimum heat flux distribution

CHAPTER 7

CONCLUSION

The Static Stress, Contact Stress and Thermal Analysis is carried out for the mentioned materials and the following conclusions were made from the work.

1. As the Static Load is applied along the vertical direction all the materials showed almost similar equivalent stresses. However, Deformations showing a difference that is negligible.
2. The Temperatures distribution for all the materials is within the limit except Nylon.
3. The Heat Flux distribution for Nylon is less when compared with other materials.
4. Based on this work, among all the materials CFRP can be suggested as the best replacement for the Structural steel material in order to improve high Power to Weight ratio.

CHAPTER 8

REFERENCES

[1] S.Mahendran , K.M.eazhil and L.Senthil Kumar of SNS College of Engineering ”Design and Analysis of Composite Spur Gear ”,IJRSI volume I, Issue VI, November 2014.

[2] Slobodan Garic , Lozica Ivanovic , Blaza Stojanovic , Slavica Miladinovic , Vladimir Milovanovic of University of East Sarajevo "Numerical Analysis of Alumiinium Composite Cylindrical Gears", Conference on Mechanical Engineering Technology and Applications.

[3] Tushar Jain , Shubham S.Jain , Shabnam Kazi , Vishal Jain , S R Dulagnge "Bending Stress and Thermal Analysis of Spur Gear by using FEM" , International Journal Of Research Publications in Engineering and Technology,IJRPET, National Conference on Innovative Trends in Engineering and Technology-2017

[4] Anand Mohan Singh , Megha Bhawsar , Neeraj Kumar Nagayach of OIST , Bhopal "Structure Analysis , Contact Stress Analysis and Fatigue Analysis of Spur Gear Assembly by Using FEM" ,International Journal Online of Science , IJO-Science , Volume 4, Issue 4,April-2015.

[5] Ismail Ali Abdul Aziz, Daing Mohamad Nafiz Daing Idris and Wafiuddin Mohd Ghazali of University Malaysia Pahang "Investigating bending strength of Spur Gear: A review", MATEC Web of conference

[6] Mr.Kailas pathade, Prof. Santosh MaliPatil of Maratha Mandal Engineering College "Determination of True Bending Stress in Spur Gear" International Research Journal Of Engineering and Technology , IRJET , Volume 3 , Issue 11, November-2016

[7] Shiferaw Damtie , Daniel Tilahun of School of Mechanical & Industrial Engineering, AAU , "Contact Stress Analysis of Involute Spur Gear by Finite Element Method", Journal Of EEA , Volume 32 , December 2014.