RESEARCH ARTICLE OPEN ACCESS

# Static Structural analysis of gear tooth

Pravin B. Sonawane[1], P.G.Damle[2]
[1]B.E. Mechanical, SSBT's COE Bambhori, Jalgaon
2(Associate Prof.) Department. of Mechanical Engineering ,COE , Bambhori, Jalgaon.

## Abstract:

This study investigate the characteristics of a gear system including contact stresses, bending stresses, and the transmission errors of gears in mesh. Gearing is one of most critical component in mechanical power transmission systems. The bending stresses in the tooth root were examined using 3D model.

Current methods of calculating gear contact stresses use Hertz's equations and Lewis Equation which were originally derived for contact between two cylinders. To enable the investigation of contact problems with FEM, the stiffness relationship between the two contact areas is usually established through a spring placed between the two contacting areas. This can be achieve by inserting a contact element in between the two areas where contact occurs. The results of the three dimensional FEM analysis from ANSYS are presented. These stresses were compared with theoretical values. Both results agree very well. This indicates that the FEM model is accurate.

This report also considers the variations of the whole body stiffness arising from gear body rotation due to bending deflection, shearing displacement and contact deformation. Many different positions within the meshing cycle were investigated. Investigation of contact and bending stress characteristics of spur gears continues to be immense attention to both engineers and researchers in spite of many studies in the past.

This is because of the advances in the engineering technology that demands for gears with ever increasing load capacities and speeds with high reliability, the designers need to be able to accurately predict the stresses experienced the stresses by the loaded gears.

*Keywords* — **Spur Gear Tooth, Ansys 14.5 Workbench and classic, Lewis Equation.**

**INTRODUCTION:**

Spur gears are the most common means of transmitting power in the modern mechanical world. They vary from tiny size used in the watches to the large gears used in marine speed reducers; bridge lifting mechanism and railroad turn table drivers. They for vital elements of main and ancillary mechanism in many machine tools, rolling mills, hoisting and transmitting machinery and marine engines etc.

The four major failure modes in gear systems are tooth bending fatigue, contact fatigue, surface wear and scoring. Two kinds of teeth damage can occur on gears under repeated loading due to fatigue; namely the pitting of gear teeth flanks and tooth breakage in the tooth root. Tooth breakage is clearly the worst case of damage, since the gear could have seriously hampered operating condition. Because of this, the stress in the tooth should always be carefully studied in all practical gear application. The fatigue process leading to tooth breakage is divided into crack initiation and crack propagation period. However, the crack initiation period generally account for the most of service life, especially in high cycle fatigue.

Spur gears are very useful in numerous applications. Not only can they transfer velocity and torque one shaft to other shaft, but, by using different size gears, they can alter the ratio between velocity and torque as they transfer them; a gear with many teeth driving a gear with fewer teeth will have less torque, but greater velocity and vice versa. Unfortunately, spur gears require a very specific shape for their teeth to work smoothly. Even a simple mockup gear would require a complex surface in order to function properly. Without the calculations required to create these surfaces, two gears would not mesh together smoothly, making it difficult to test the gears. But mockup does not need to work for a long period of time; it can be made of lighter, easily cut materials. These lighter materials could be handle by a laser cutter if suitable instructions were developed so that the proper shape of teeth could be computed.

The tooth of spur gear is based on a mathematical shape as an involute. Since each tooth can be described by a series of mathematical equations, it is possible to define a gear in terms of a few key parameters, such as the number of teeth and diameteral pitch. These parameters make it easy to tell if two gears can mesh together. Similarly by specifying the parameters first, it would be simple to design a gear for any given applications from scratch.

Because spur gear is essentially two dimensional shapes they could be cut out quickly using a laser cutter. Laser cutters use a laser beam to slice two dimensional shapes out of flat material, so the silhouette of a spur gear would be easy to make using a laser cutter. Unfortunately, most laser cutter do not have built in software's to cut gears; They are driven by a series of simple move-to and draw-to commands, tracing out straight lines or elliptical arcs across the material. Therefore, an involute must first be converted into approximations using these simple commands in order for the cutter to understand it. If a laser cutter is to be useful in creating mockups, it must be able to cut these gear designs quickly and easily.

Figure 1 Gear teeth meshing.[1]

**SPUR GEAR TERMINOLOGY:**

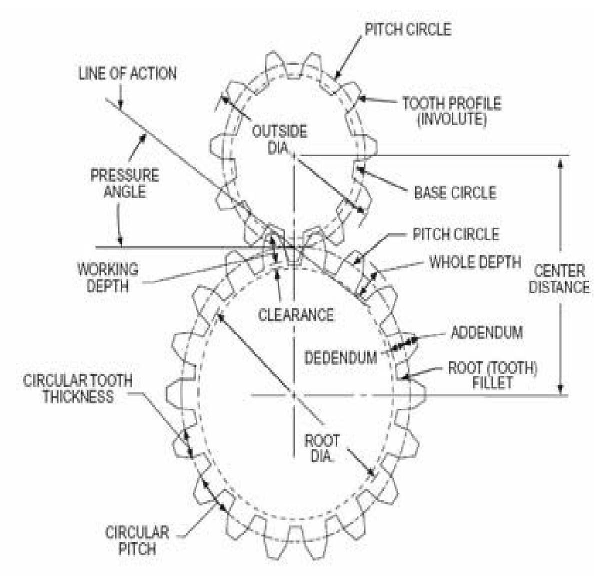



Figure 3.1 Gear terminology [1]

**DEFINITIONS:-**

1) Module: - It is defined as the ratio of diameter to the number of teeth.
   $M = d/Tg$
2) Face width (b):- It is the width along the contact surface between the gears.
3) Addendum: - The radial distance between the pitch circle and the top land of the gear is called the addendum.
4) Dedendum: - The radial distance between the pitch circle and the bottom land of the gear is called dedendum.

5) Pitch circle: - It is an imaginary circle which by pure rolling action would give the same motion as the actual gear.
6) Root circle: - It is the circle drawn through the bottom of the tooth.
7) Pressure angle: - It is the angle between the common normal to two gear teeth at the point of contact and the common tangent at the pitch point. The standard pressure angles are 14.5 and 20.
8) Diametral pitch: - it is the ratio of number of teeth to the pitch circle diameter.

$$p = T/D$$

**DESIGN PARAMETERS:**

$Kv$ = Velocity factor

$K$ = Structural stiffness

$U$ = Displacement vector

$F$ =Applied load vector

$Pmax$ = Maximum contact stress

$Dp$ = Pinion pitch diameter

$Dg$ = Gear pitch diameter

$Fi$ =Load per unit width

$\Phi$ =Pressure angle

$E$ = Young's modulus

$b$ = face width

$pd$ = Diametric pitch

$Y$ = Lewis form factor

$Ks$ = Size factor

$Ft$ = Normal tangential load

**MATHEMATICAL CALCULATIONS:**

**PROBLEM:-**

Calculate the power that can be transmitted safely by a pair of spur gears with the data given below. Calculate the power and the bending stress induced in the two wheels when the pair transmits that power. **[1]**

The theoretical design calculations are performed using the input parameters given below

No. of Teeth in the Pinion = 20

No. of Teeth in the Gear = 80

Module m = 4 mm

Width of Teeth = 60 mm

Speed of Pinion N = 400 rpm

Tooth profile = 20 degree involute

Service factor = Cs = 0.8

Velocity factor = Kv = 3

Allowable bending strength of material

= 200 Mpa, for pinion

= 160 Mpa, for Gear

**SOLUTION:-**

Lewis form factor = $0.154 - (0.912/T)$

Velocity factor $Cv = 3/(3+v)$

$Dp = m * Tp$ = 4 * 20 = 80 mm

$Dg = m * Tg$ = 4 * 80 = 320 mm

Addendum = m = 4 mm

Dedendum = $1.25 * m$ = 1.25* 4 = 5 mm

Minimum clearance = $0.25 * m$ = 0.25 * 4 = 1 mm

Tip circle dia. Of pinion = $Dp + (2*m)$ = 88 mm

Root circle dia. Of Pinion = $Dp - (2 * m)$ = 72 mm

Tip circle dia. Of Gear = $Dg + (2 * m)$ = 328 mm

Root circle dia. Of Gear = $Dg - (2*m)$ = 312 mm

$V = 3.14* Dp*Np/ (60*1000)$

= 3.14*80*400/(60*1000)

= 1.67 m/s

Velocity factor $Cv = 3 / (3+ 1.67)$

= 0.642

Lewis form factor for pinion =

$Yp = 0.154 - (0.912/20)$

= 0.1084

Lewis form factor for Gear =

$Yg = 0.154 - (0.912/80)$

= 0.1426

Therefore tangential force transmitted is given by

$Ft = L * Cv * 3.14 * b * m * Yg$

$= 200*0.642*3.14*60*4*0.1426$

= 8382.40 N

Now power transmitted for the given force is

$P = Ft *1.67$ = 13998.60

=13.99 kw

For calculating bending stress for gear, the Lewis equation is

$\sigma t = Ft/(Kv*m*b*Yp)$

$\sigma t$ = 8382.40/(3*4*60*0.1084)

$\sigma t$ = 107.40 Mpa

And bending stress for pinion is calculated using

$\sigma t = Ft/(Kv*m*b*Yg)$
$= 8382.4/(3*4*60*0.1420)$
$= 81.98$ Mpa

**MATERIAL PROPERTIES:-**

| Properties | Gear |
|---|---|
| Material Name | Structural Steel |
| Youngs Modulus | 2E5 Mpa |
| Yield strength | 250 Mpa |
| Poissons Ratio | 0.3 |
| Density | 7850 kg/m3 |

**FINITE ELEMENT ANALYSIS:**

The basic concept in FEA is that the body or structure may be divided into smaller elements of finite dimensions called “Finite elements”. The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called “Nodes” or “Nodal Points”. Simple functions are chosen to approximate the displacement over each finite element. Such assumed functions are called “shape functions”. This will represent the displacement within the element in terms of the displacement at the nodes of the element.

The FEM is a mathematical tool for solving ordinary and partial differential equations. Because it is a numerical tool, it has the ability to solve the complex problem that can be represented in differential equations form. The applications of FEM are listless as regards the solution of practical design problems. Due to high cost of computing power of years gone by, FEA has a history of being used to solve complex and cost critical problems. Classical methods alone usually cannot provide adequate information to determine the safe working limits of a major civil engineering construction or an automobile or aircraft.

In the recent years, FEA has been universally used to solve structural engineering problems. The departments, which are heavily relied on this technology, are the automotive and aerospace industry. Due to the need to meet the extreme demands for faster, stronger, efficient and lightweight automobiles and aircraft, manufacturers have to reply on this technique to stay competitive.

FEA has been used routinely in high volume production and manufacturing industries for many years, as to get a product design wrong would be detrimental. For example, if a large manufacturer had to recall one model alone due to a hand brake design fault, they would end up having to replace up to few millions of hand brakes. This will cause a heavier loss to the company.

**STATIC ANALYSIS:-**

There are two types of static analysis

1) Linear static analysis:

Linear means straight line. In this analysis the ratio stress to strain is linear or straight line. But in real life after crossing yield point material follows nonlinear curve but software follows same straight line. Component brake into two separate pieces after crossing ultimate stress but software based analysis never shows failure in this fashion. It shows single unbroken part with red colour zone at the location of failure. Analyst has to conclude whether the component is safe or failed by comparing the maximum stress value with yield or ultimate stress.

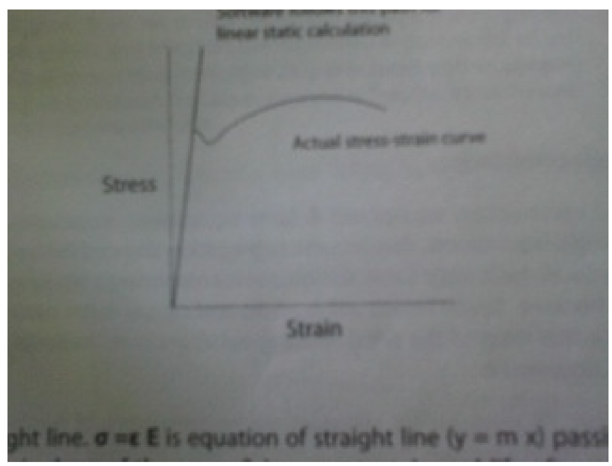



**Fig. 5.1 Stress and Strain curve**

There are two conditions for static analysis

Force is static. i.e. no variation with respect to time(dead weight).

Equilibrium condition: - summation of forces and moments in x,y and z direction is zero. FE model fulfils this condition at each and every nodes.

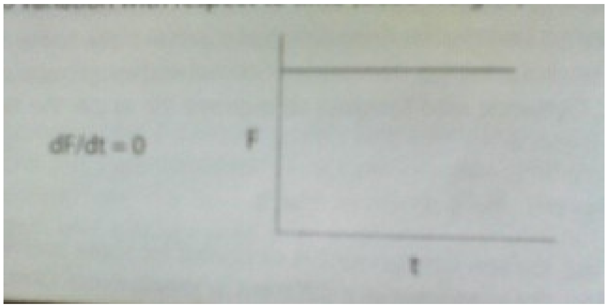



**Fig. 5.2 static force**

2) Non-linear static analysis:

In nonlinear analysis stress Vs strain curve is nonlinear.

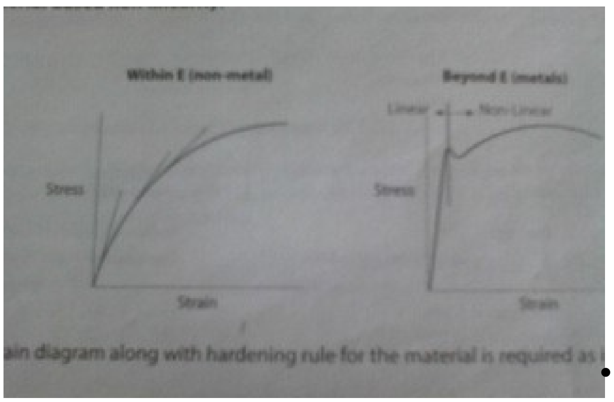



**Fig.5.3 stress and strain curve of metals and nonmetals**

**WHY FEA?**

FEA used in problems where analytical solutions are not easily obtained. Mathematical expressions required for solutions not simple because of complex geometry, loading and material properties.

Analytical methods involve solving for entire system in one operation. FEA involving defining equations for each element and combining to obtain system solution.

**BASIC STEPS INVOLVED IN FEA:**

Basic steps: -

**Discretization of domain:-**

All real life objects are continuous. Means there is no physical gap between any two consecutive particles. As per material science any object is made of small particles, particles of molecules, molecules of atom and so on. Hence the task is to divide the continuous object into number of subdivisions called Element. Based on continuum it can be divided into line or area or volume elements.

**Application of Boundary conditions:-**

From the physics of the problem we have to apply the field conditions i. e. loads and constraints, which will help us in solving for the unknowns.

**Assembling the system equations:-**

This involves the formulations of respective characteristics (stiffness in case of structural) equation of matrices and assembly.

**Solution for system equation:-**

Solving for the equations to know the unknowns. This is basically the system of matrices which are nothing but a set of simultaneous equations are solved.

**Viewing the results:-**

After the completion of the solution we have to review the required results. The first two steps of the above said process is known as pre-processing stage, third and fourth is the processing stage and final stage is known as post-processing stage.

**What is an element?**

Element is an entity, into which a system under study can be divided into. An element definition can be specified by nodes. The shape (area, length, and volume) of the element depends upon the nodes with which it is made up of.

- **What are Nodes?**

Nodes are the corner points of the element. Nodes are independent entities in the space. These are similar to points in geometry. By moving a node in space an element shape can be changed.

**TYPES OF FINITE ELEMENTS:**

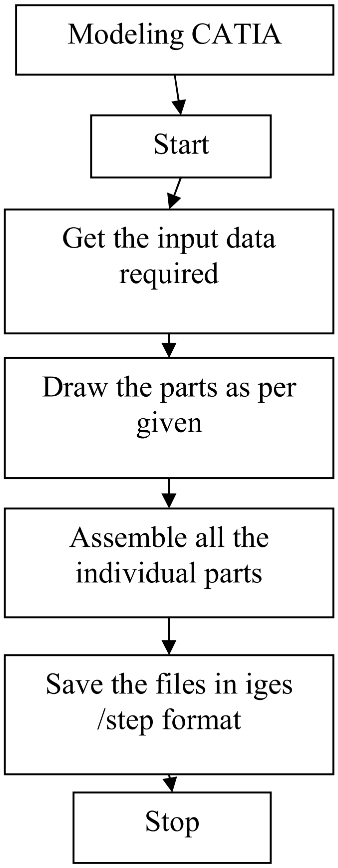
1-D Element: -( Line Elements: )

2-D Element:-

3-D Element: -

**CREATING A SOLID MODEL:**

Modeling provides the design engineer comfortable modeling techniques such as sketching, feature based modeling, and dimension driven editing. An excellent way to begin a design concept is with a sketch. When you use a sketch, a rough idea of the part becomes represented and constrained, based on the fit and function requirements of your design. In this way, your design intent is captured. This ensures that when the design is passed down to the next level of engineering, the basic requirements are not lost when the design is edited.




**PARAMETRIC MODELING OF GEAR USING CATIA V5**

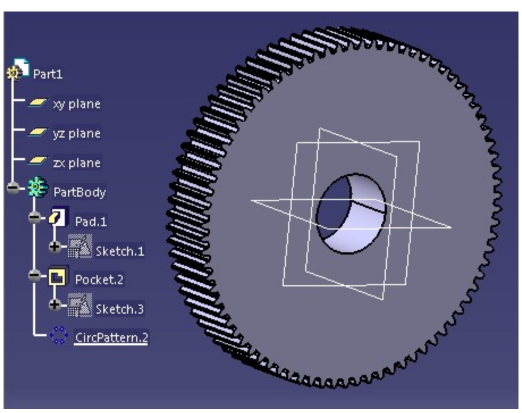



**Figure 7.1 3D model of gear**

The strategy you use to create and edit your model to form the desired object depends on the form and complexity of the object. You will likely use several different methods during a work session. The next several figures illustrate one example the design process, starting with a sketch and ending with a finished model. First, you can create a sketch "outline" of curves.

Then you can sweep or rotate these curves to create a complex portion of your design.

**SPUR GEAR ANALYSIS:**

Now, the 3D which was created in CATIA is imported in ANSYS workbench 12 for stress analysis. It is done by saving drawing in STP or IGS file format in CATIA. After the model is imported in ANSYS workbench 12, as the both teeth are already in contact, our main purpose is to find the root bending stress. This is done by using following steps in ANSYS workbench 12.

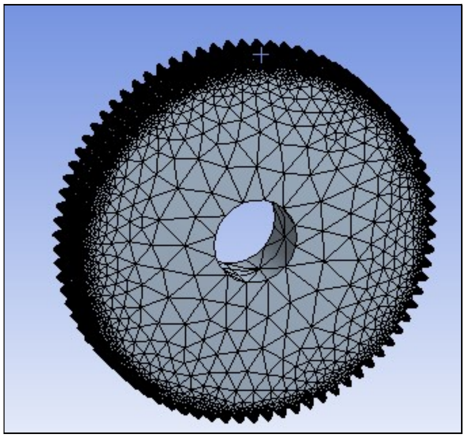
The objective of the analysis is to perform structural static analysis on the gear by applying tangential load and examine the deflections and stresses.

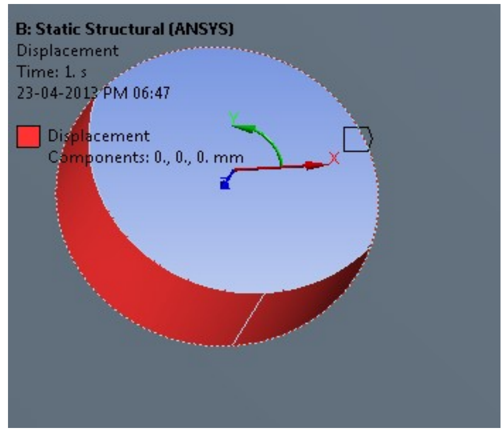
The 3-D model of the spur gear in CATIA converted it into “iges” and “step” file. And then iges file is then imported into ansys workbench.

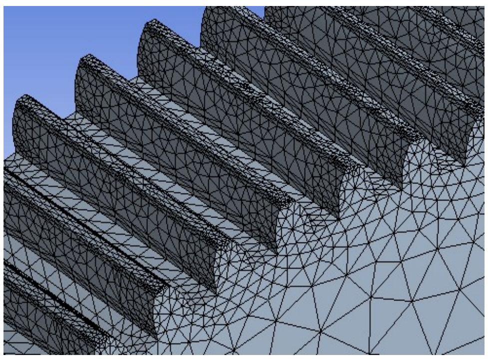
**3-DIMENSIONAL ANALYSIS OF SPUR GEAR**: **-** For imported 3-dimensional geometry, firstly we select 3-D and static structural analysis from menu and connecting the geometry to the analysis tab. Then next we enter Young’s modulus and Poisson’s ratio of the material.

**MESH GENERATION AND BOUNDARY CONDITION**:- (supports and load):- A tetrahedron solid elements is used in mesh generations. Boundary condition refers to the external load on the border of the structure. We assumed gear is with fixed support and pinion is subjected to a moment or torque along its axis with frictionless support.

**MESHING OF GEAR**:-





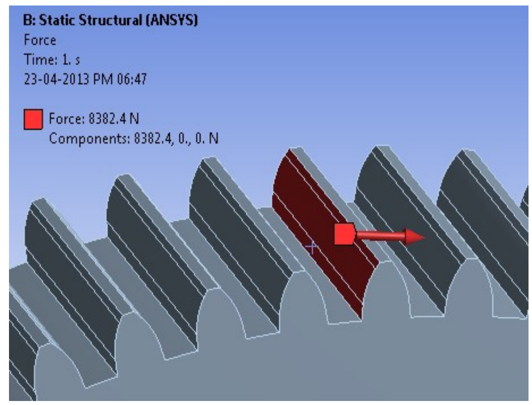
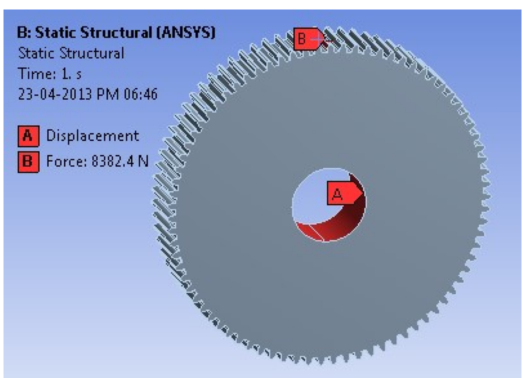


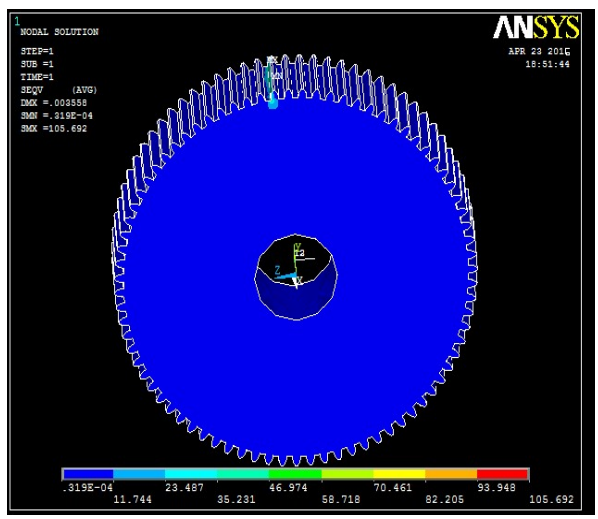

**Figure 8.1 Boundary Conditions**

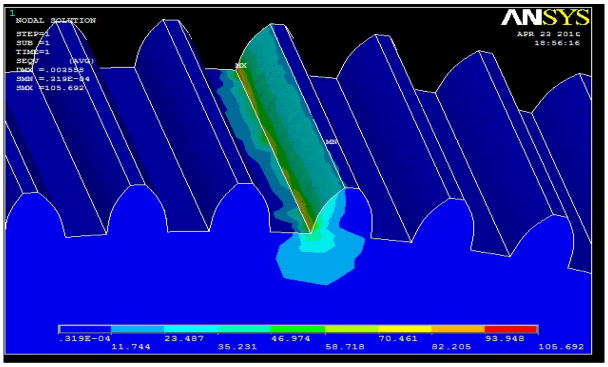
**BOUNDARY CONDITION OF GEAR:-**



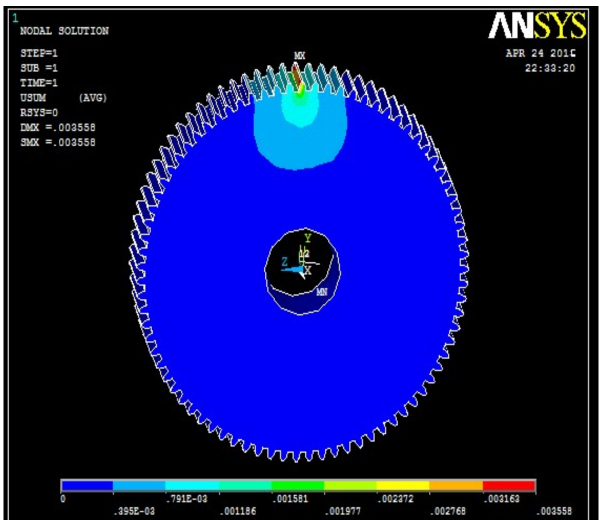

**RESULT:**

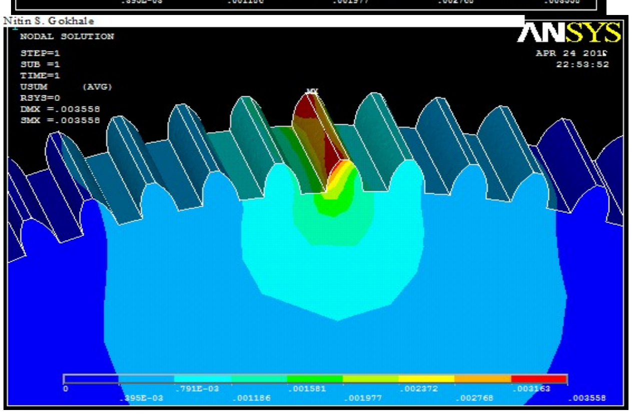
**MAXIMUM VONMISSES STRESS:-**






**DEFLECTION :-**




**CONCLUSION:**

The above study shows that the Finite Element Method can be used for bending stress analysis in a pair of gear. Bending Stress calculation is play more significant role in the design of gear. This study is shows that Lewis formula is use for calculating bending stress in a pair of gear. Theoretically result obtained by Lewis formula are comparable with Finite Element Analysis of spur gear.

Maximum von mises stress observed in structural steel gear is 105 Mpa on Ansys and 107 Mpa by theoretically.

Maximum deflection of 0.035mm observed in the gear along x-direction.

**ACKNOWLEDGEMENT:**

It gives me a great pleasure in expressing my thanks and profound gratitude to **Dr. D. S. Deshmukh** HOD, Mechanical Engineering Department, COE, Bambhori,Jalgaon for their valuable guidance and encouragement throughout. I am heartily thankful to him for his continuous suggestions and clarifying the concepts of the topic that helped me a lot during this study.

**REFERENCE:**

1) R.S.Khurmi " Machine Design".

2) Nitin S. Gokhale "Practical Finite Element Analysis".

3) Sushilkumar Tiwari and Upendra kumar Joshi "Stress Analysis Of Mating Involute Spur Gear"IJERT 2012.

4) Ansys 14.5

5) Catia v5 R20

I Am Pravin B. Sonawane , B.E. Mechanical from MSS's CET,Nagewadi,Jalna,2010.
Now Pursuing M.E. Mechanical (Machine Design) from SSBT's COE Bambhori, Jalgaon.

Guide- P.G.Damle, (Associate Prof.) Department. of Mechanical Engineering ,COE , Bambhori, Jalgaon.