ISSN 2395-1621

Modeling and Finite Element Analysis of Flywheel Ring Gear & Starter Motor Pinion

#1Parag Vyankatrao Thote, #2Dr. S.P. Deshmukh

1prgthote4@gmail.com
2suhas.deshmukh@gmail.com

1M. E. Automotive Engg. SAOE, Pune, Maharashtra, India
2Professor, Mechanical Engg. Department, SAOE, Pune,Maharashtra, India
India

ABSTRACT

Gears are the most important parts of Mechanical system they are generally used to transmission power from one shaft to another depending on application they are available with different type of tooth profiles. In all spur gears are the most preferred type of gear because of simplicity of use & manufacturing with high degree of transmission efficiency. The gears are generally fails when the working stress exceeds the maximum permissible stress and if we want to design a healthy system with defined performance efficiency through working life cycles, it is important to predict stresses developed & effectively reduce them. When concerning about starter motor, pinion is the element which damage mostly because of the unusual engagement with flywheel ring gear. In this paper an attempt is made to predict static stresses and deformation of tooth in contact, after determination of predictions the system is modified in order to study its effect on stresses developed and deformation observed over original working geometry. As the gear tooth engagement of the system is unlikely different from normal types of system of gears in mesh, the tooth tip deforms at higher rate because of abrupt contact with tooth of another gear. A professional tool (ANSYS 16.0) is used to carry finite element analysis. Most of the previous research work studied elaborates the practical ability of finite element analysis (FEA) for study of gears in mesh.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

Keywords— FEA, Static & Dynamic Stresses, Starter Motor Pinion, Ring Gear.

I. INTRODUCTION

A gear also called as cogwheel is the most important & critical element of power transmission system. Gear is a rotating cylindrical wheel having tooth cut on it, which meshes with another toothed part to transmit the power, in most cases with teeth on the one gear being of identical shape, and often also with that shape on the other gear in mesh. There are different types of gears like Spur, Helical, Worm and Bevel. They consist of a cylinder or disk with the teeth projecting radially, and although they are not straight-sided in form (they are usually of special form to achieve constant drive ratio, mainly involute), the edge of each tooth is straight and aligned parallel to the axis of rotation. These gears can be meshed together correctly only if they are fitted to parallel shafts. As the most common type, spur gears are often used because they are the simplest to design & manufacture, less costly, efficient with 98-99% operating efficiency. They are usually employed to achieve constant drive ratio. There are several stresses present in the teeth of rotating gears but out of all the stresses, root bending stress and surface contact stress calculation is the basic of stress analysis. Theoretically, for the calculation of contact stress at the surface of mating teeth, Hertz equation is used and for determining bending stress at the root of meshing gears, Lewis formula is used. In detail study of the contact stress produced in the mating gears is the most important task in design of gears as it is the deciding parameter in finding the dimensions of gear. Also the module of a gear plays an

important role in transmitting the power between two shafts. The spur gear with higher module is the best choice for transmitting large power between the parallel shafts.

A pair of teeth in action is generally subjected to two types of cyclic stresses: bending stresses inducing bending fatigue and contact stress causing contact fatigue. Both these types of stresses may not attain their maximum values at the same point of contact. However, combined action of both of them is the reason of failure of gear tooth leading to fracture at the root of a tooth under bending fatigue and surface failure, like pitting or flaking due to contact fatigue. In addition there may be surface damage associated seizure of surfaces due to poor lubrication and overloading. The seizure of surfaces leading to welding is usually prevented by proper lubrication so that there is always a very thin film of lubricant between a pair of teeth in motion. However the fracture failure at the root due to bending stress and pitting and flaking of the surfaces due to contact stress cannot be fully avoided. These types of failures can be minimized by careful analysis of the problem during the design stage and creating proper tooth surface profile with proper manufacturing methods. In spite of all the cares, these stresses are sometimes very high either due to overloading or wear of surfaces with use and need proper investigation to accurately predict them under stabilized working conditioned so that unforeseen failure of gear tooth can be minimized.

The increasing demand for quiet power transmission in machines, vehicles, elevators and generators, has created a growing demand for a more precise analysis of the characteristics of gear systems. In the automobile industry, the largest manufacturer of gears, higher reliability and lighter weight gears are necessary as lighter automobiles continue to be in demand. In addition, the success in engine noise reduction promotes the production of quieter gear pairs for further noise reduction. Designing highly loaded spur gears for power transmission systems that are both strong and quiet requires analysis methods that can easily be implemented and also provide information on contact and bending stresses, along with transmission errors. The finite element method is capable of providing this information, but the time needed to create such a model is large. In order to reduce the modeling time, a preprocessor method that creates the geometry needed for a finite element analysis may be used, such as that provided by CATIA, it can generate models of three-dimensional gears easily.

Gears analyses in the past were performed using analytical methods, which required a number of assumptions and simplifications. In general, gear analyses are multidisciplinary, including calculations related to the tooth stresses and to tribological failures such as like wear or scoring. In this thesis, static contact and bending stress analyses were performed, while trying to design spur gears to resist bending failure and pitting of the teeth, as both affect transmission error. As computers have become more and more powerful, people have tended to use numerical approaches to develop theoretical models to predict the effect of whatever are studied. This has improved gear analyses and computer simulations. Numerical methods can potentially provide more accurate solutions since they normally require much less restrictive assumptions. The model and the solution methods, however, must be chosen carefully to ensure that the results are accurate and that the computational time is reasonable. The finite element method is very often used to analyze the stress state of an elastic body with complicated geometry, such as a gear. There have been numerous research studies in the area.

- **Study of Literature**

Quasi static finite element analysis was carried out for NCR & HCR gears with fixed module, centre distance & gear ratio. Here the increasing contact ratio is obtained by increasing the addendum factor from 1.0 to 1.25 m. Hence a contact ratio of more than 2.0 was achieved for the same number of tooth. Two dimensional deformable body contact models for both HCR gear & NCR gears were created using the ANSYS-APDL loop program. Various parameters such as load sharing ratio, bending stress & contact stress were evaluated and compared over the path of contact. The maximum bending stress for a HCR gear is 18% less & contact stress is 19% less than of a NCR gear for the pair of same module & fixed center distance. Hence the load carrying capacity of the HCR gear is 18% more than the NCR gear designed for the same weight, fixed module & same centre distance of gear pair [1]. In this paper researchers studied the contact stresses among the Spur gear pair & Helical gear pair, under static condition by using 3D finite element model. The Helical gear pair on which the analysis was carried out were 0° ,5°, 15°, 25° helical gear set. During analysis FE gear model was verified with Hertz/AGMA equation for zero coefficient of friction. The FE model of gear pair are compatible in evaluating the contact stresses & the results obtained are in good agreement with analytical calculations. For the spur gear pair the increase in contact stress with the increase in coefficient of friction was about 10% [2].

II. SYSTEM MODEL

In this study the system of gears used is consists of 8 tooth pinion & 126 tooth ring gear, both gears used are of spur type. Material used while carrying finite element analysis is structural steel. This study mainly works on the behaviour of gear assembly, the starter motor pinion & flywheel ring gear. As like other gear assembly, tooth's of mating gears are always in constant mesh, but in case of our system the starter pinion will move forward and made contact with ring gear. Because of the high speed rotation of the pinion gear when it come in contact with ring gear it impacts on initial face of gear tooth i.e. gear tip, this abrupt contact between two tooth wear of tooth tip comes into picture. The stresses developed at root of the tooth are a point of interest in this work. When the tooth comes in contact the contact pressure varies with respect to contact surface increases.

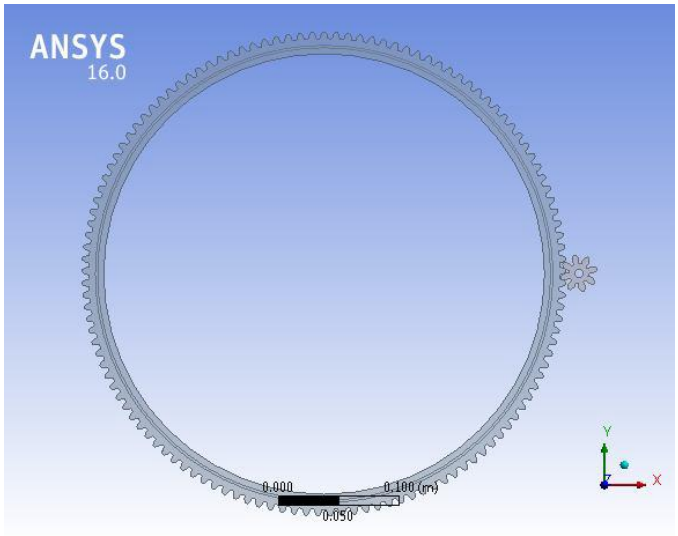


Figure1. Ring Gear and Pinion

- **Gear selection criteria**

Since there are number of machines that have applications for gears, choosing the right type of gear for the suitable application is quite a difficult task. In most cases the geometric arrangement of the apparatus that needs the gear drive will decide the gear selection. If the gears are to be on parallel axes, then spur or helical gears are the ones to be used. Bevel and worm gears can be used if the axes are at right angles but are not suitable for parallel axes drives. Type of gear used depends on the application and design requirements. For the purpose of this research only spur gear design and geometry will be considered because in case of starter pinion and ring gear spur is commonly used type of gear. This is mainly due to the fact that spur gears are the simplest form of gear, and all other gears can be derived or designed by starting with the general spur gear shape. Spur gears are also very commonly used in many machines and are wide spread in all aspects of engineering. The general gear selection criteria can be summarized as in the table below;

TABLE I
SELECTION OF GEARS ON TYPE OF SERVICE

Parallel axis	**Intersecting axis**	**Non-intersecting non-parallel axis**
Spur Gear	Straight Bevel Gear	Worm Gear
Helical Gear	Spiral Bevel Gear	

- **Introduction to Finite Element Analysis**

In finite element analysis the continuum is divided into a finite numbers of elements, having finite dimensions and reducing the continuum having infinite degrees of freedom to finite degrees of unknowns. It is assumed that the elements are connected only at the nodal points. The accuracy of solution increases with the number of elements taken. However, more number of elements will result in increased computer cost. Hence optimum number of divisions should be taken. In the element method the problem is formulated in two stages:

A) The element formulation

It involves the derivation of the element stiffness matrix which yields a relationship between nodal point forces and nodal point displacements.

B) The system formulation

It is the formulation of the stiffness and loads of the entire structure.

Basic steps in the finite element method;

- Discretization of the domain

The continuum is divided into a no. of finite elements by imaginary lines or surfaces. The interconnected elements may have different sizes and shapes. The success of this idealization lies in how closely this discretized continuum represents the actual continuum. The choice of the simple elements or higher order elements, straight or curved, its shape, refinement are to be decided before the mathematical formulation starts.

- Identification of variables

The elements are assumed to be connected at their intersecting points referred to as nodal points. At each node, unknown displacements are to be prescribed. They are dependent on the problem at hand. The problem may be identified in such a way that in addition to the displacement which occurs at the nodes depending on the physical nature of the problem.

- Choice of approximating functions

After the variables and local coordinates have been chosen, the next step is the choice of displacement function, which is the starting point of mathematical analysis. The function represents the variation of the displacement within the element. The shape of the element or the geometry may also approximate.

- Formation of element stiffness matrix

After the continuum is discretized with desired element shapes, the element stiffness matrix is formulated. Basically it is a minimization procedure. The element stiffness matrix for majority of elements is not available in explicit form. They require numerical integration for this evaluation.

- Formation of the overall stiffness matrix

After the element stiffness matrix in global coordinates is formed, they are assembled to form the overall stiffness matrix. This is done through the nodes which are common to adjacent elements. At the nodes the continuity of the displacement functions and their derivatives are established.

- Incorporation of boundary conditions

The boundary restraint conditions are to be imposed in the stiffness matrix. There are various techniques available to satisfy the boundary conditions.

- Formation of the element loading matrix.

The loading inside an element is transferred at the nodal points and consistent element loading matrix is formed.

- Formation of the overall loading matrix

The element loading matrix is combined to form the overall loading matrix. This matrix has one column per loading case and it is either a column vector or a rectangular matrix depending on the no. of loading conditions.

- Solution of simultaneous equations

All the equations required for the solution of the problem is now developed. In the displacement method, the unknowns are the nodal displacement. The Gauss elimination factorization is most commonly used methods.

- Calculation of stresses or stress resultants

The nodal displacement values are utilized for calculation of stresses. This may be done for all elements of the continuum or may be limited only to some predetermined elements.

III.FINITE ELEMENT ANALYSIS OF SYSTEM

This topic will give detailed information about finite elemental analysis of the system, ANSYS is a package used as analysis tool. Previous work by different researchers shows practical ability of ANSYS for carrying gear contact analysis. The material used is Structural Steel having allowable Compressive Yield strength limit of 2.5e+008 (Pa) so it will be a point of interest to maintain stress level below allowable stress limit of the material. While carrying finite element analysis 6 cases with different fillet radius are evaluated. The original system analyzed consists of 8 tooth pinion & 126 tooth ring gear. In order to predict extreme conditional working performance of the system only one number of tooth in contact is considered. For determination of static condition i.e. deformation & equivalent stress the "Transient Structural" analysis module is used.

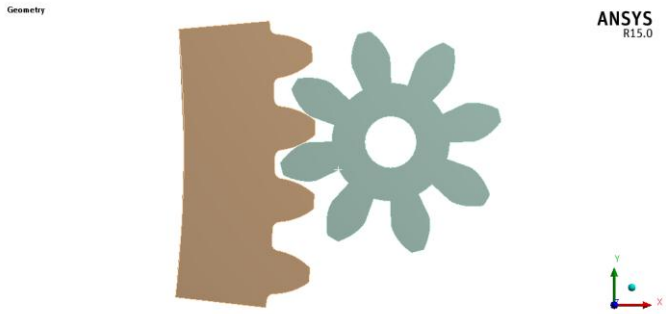


Figure 2. Working Geometry

For the ease of working and reducing analysis time the number of tooth on ring gear are reduced to 4 tooth only, this action is done by using "slice" command available in ANSYS design modular.

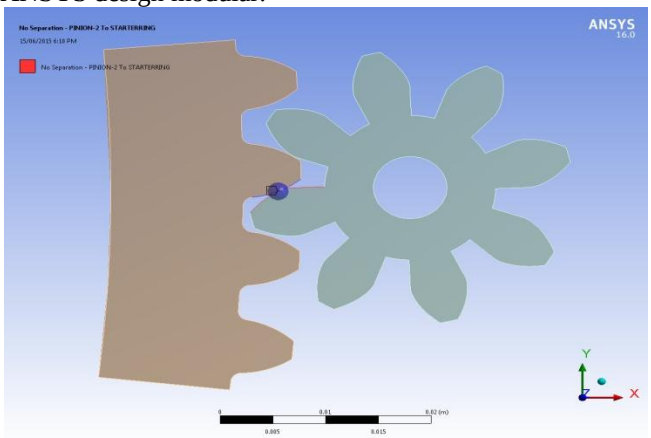


Figure 3. Contact between to gears

The type of contact used is frictionless no separation type, with added pinion as a contact body & ring gear as a target body.

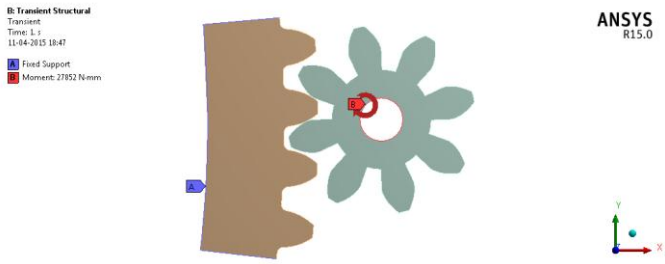
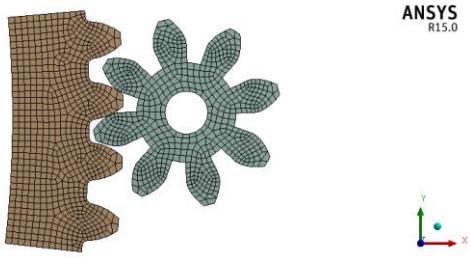


Figure 4. Fix support and moment

The ring gear body is considered as a fix support and moment of "-27.852 N-m" is added along Z-Axis. We are running this assembly at 1200RPM and the rated power output of Starter Motor is 3.5KW.



. Figure 5. Meshing of geometry

After meshing this contact assembly number of Nodes and Elements formed are 4613 & 4306 respectively.

To find maximum deformation & equivalent stress generated analysis of 6 different cases with varying fillet radius are evaluated. The modification range is from 0.5mm to 1.5mm is compared with the results obtain from working geometry. All of the cases studied are detailed below:

- **Determination of deflection & stress in working geometry**

In 1st case the system analyzed is a working geometry consisting of 8 tooth pinion & 4 tooth ring gear. The results obtained are deformation (0.0102mm) and equivalent stress (104.53MPa).

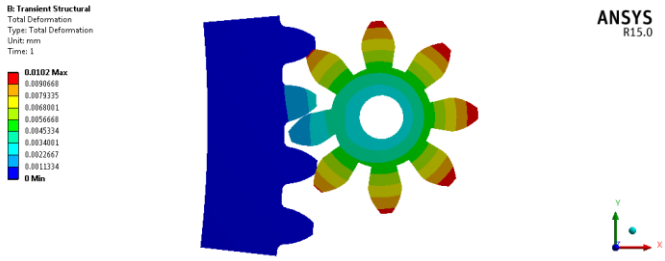


Figure 6. Deflection for working geometry

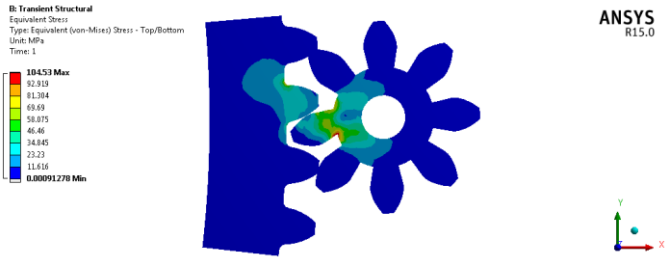


Figure 7. Equivalent stress for working geometry

- **Case 1 (root fillet 0.5mm)**

In 2nd case the root fillet of 0.5mm added to pinion and obtained deformation (0.0097446mm) and equivalent stresses (150.51MPa).

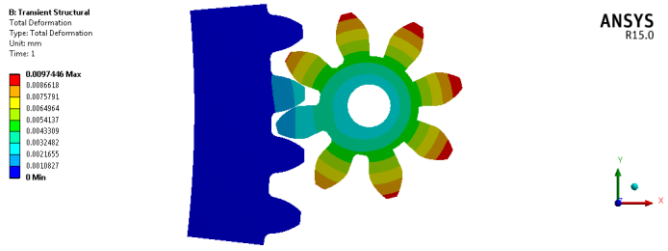


Figure 8. Deformation for root fillet 0.5mm

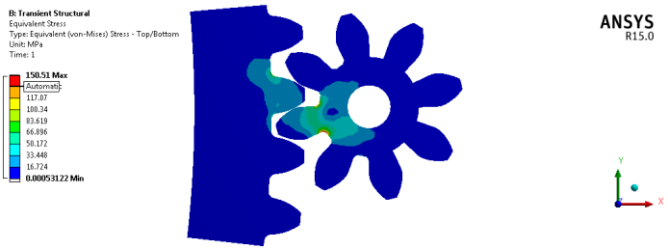


Figure 9. Equivalent stress for root fillet 0.5mm

- **Case 2 (root fillet 0.75mm)**

While analyzing 3rd case the root fillet of 0.75mm is added, the deformation observed (0.0096591mm) and equivalent stress (170.70MPa).

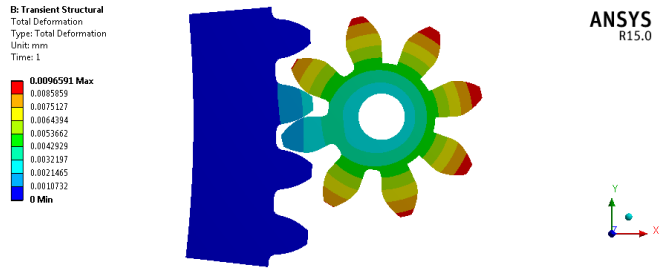


Figure 10. Deformation for root fillet 0.75mm

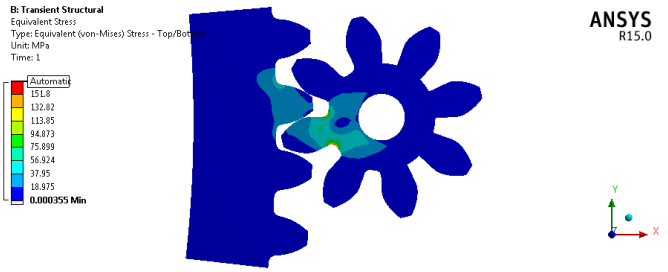


Figure 11. Equivalent stress for root fillet 0.75mm

- **Case 4 (root fillet 1mm)**

While considering 4th case the root fillet of 1mm is added, the deformation (0.0096591mm) and equivalent stress (170.77MPa) is observed.

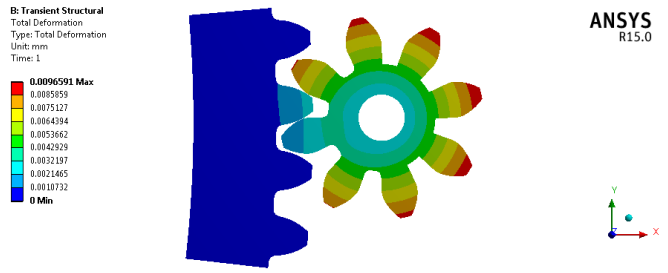


Figure 12. Deformation for root fillet 1mm

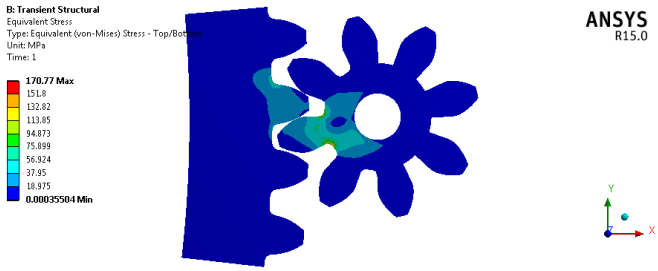


Figure 13. Equivalent stress for root fillet 1mm

- **Case 5 (root fillet 1.25mm)**

In 5th case with added 1.25mm fillet the deformation (0.009326mm) and equivalent stress (161.18MPa) are observed.

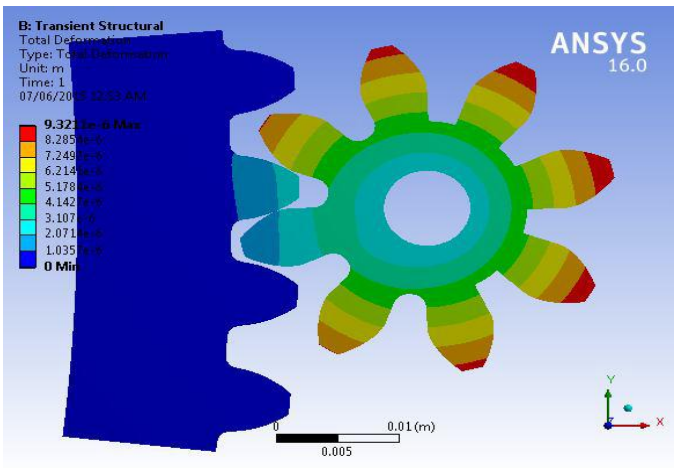


Figure 14. Deformation for root fillet 1.25mm

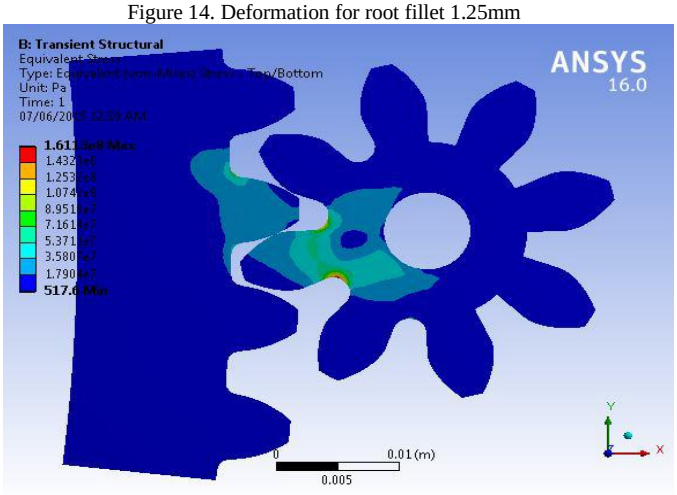


Figure 15. Equivalent stress for root fillet 1.25mm

- **Case 6 (fillet radius 1.5mm)**

The 6th case is analyzed by adding 1.5mm root fillet, the deformation (160.91mm) and equivalent stress (0.0096931MPa) are observed.

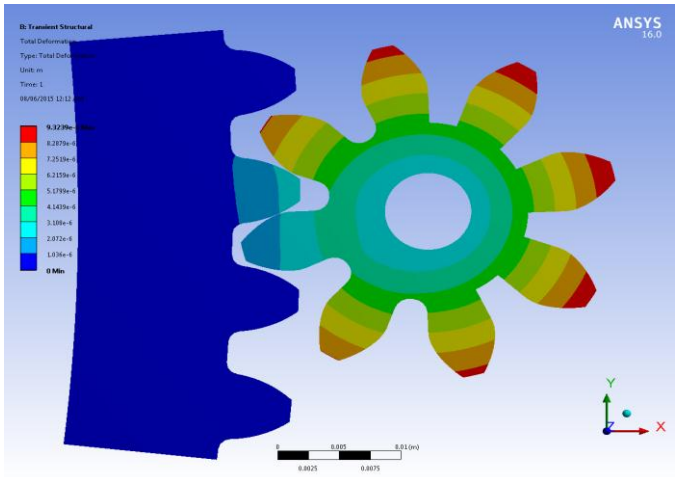


Figure 16. Deformation for root fillet 1.5mm

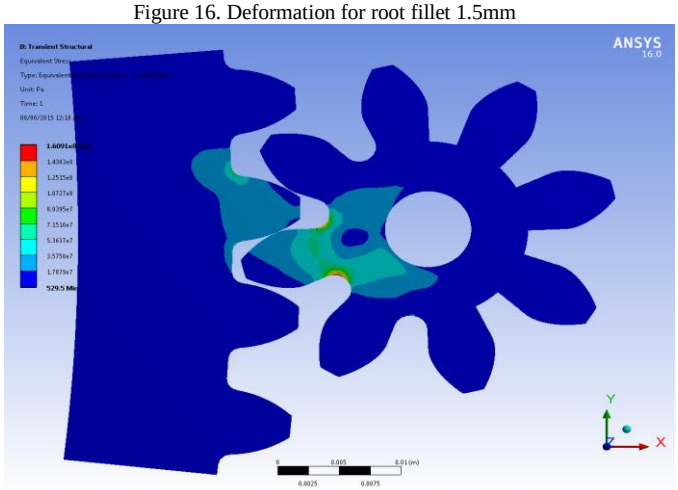


Figure 17. Equivalent stress for root fillet 1.5mm

IV.RESULT & DISCUSSION

The system is studied in ANSYS for deformation & Equivalent Stress, while analyzing it for Equivalent Stress is observed that, the stresses generated are slightly higher than the stress observed in working geometry i.e. 104.53MPa, while it will go on continuously increasing up to the addition of root fillet radius by 1mm, means stresses observes are 170.77MPa. When we continuously go on adding root fillet radius as 1.25mm equivalent stresses are observed 161.18MPa which is slightly lower than 170.77MPa. Same further lowering in Equivalent stress value is observed 160.91MPa when we add root fillet by 1.5mm. All the stresses observed in each case are lower than the allowable Compressive Yield strength limit of (2.5e+008Pa) the material used i.e. Structural Steel. When deformation comes into picture, for working geometry the deformation observed is 1.0102mm, when we go on increasing root fillet the deformation of system is go on slightly decreasing i.e. for 1.5mm added root fillet radius the deformation is 0.0093239mm. Main purpose of this work is to reduce wear of the pinion i.e. reducing deformation.

TABLE III
RESULTS OF STRESSES & DEFORMATIONS

Sr. No.	Added Root Fillet (mm)	Stress (MPa)	Deformation (mm)
1	Working geometry	104.53	0.0102
2	0.5	150.51	0.0097446
3	0.75	170.70	0.0096591
4	1.0	170.77	0.0096591
5	1.25	161.18	0.009326
6	1.5	160.91	0.0093239

V. CONCLUSION.

The starter motor pinion & ring gear are in unusual contact. The transient structural analysis is carried in order to determine stresses & deformation in the system. We are considering only one number of tooth in fully contact to predict extreme working simulation, whole load is absorb by single tooth. The stresses generated are within the maximum permissible limit of material i.e. 2.5e+008Pa. The deformation is reduced from 0.0102mm to 0.0093239mm. The best root fillet radius observed is 1.5mm in which equivalent stress generated 160.91MPa & deformation 0.0093239mm. FEA is a good tool for analyzing contact problems in which Mathematical formulation of the system consists is quite difficult and need to have number of assumptions.

REFERENCES

[1]. M. Rameshkumar, G. Venkatesan, P. Shivkumar, DRDO, Ministry of Defence, “Finite Element Analysis of High Contact Ratio Gear” AGMA Technical paper**,** ISBN: 978-1-55589-981-3

[2]. Santosh S. Patil, Saravanan Karuppanan, Ivana Atansovska, Azmi Abdul Wahab, “Contact stress analysis of Helical gear pair including frictional coefficient”, International journal of Mechanical Sciences 85(2014) 205-211

[3]. Vivek Karaveer, Ashish Mogrekar & T. Preman Reynold Joseph, “Modeling & finite element analysis of spur gear”, International journal of current Engineering & technology, ISSN: 2277-4106

[4]. Mrs. Shinde S.P., Mr. Nikam A.A., Mr. Mulla T.S., “Static analysis of spur gear using finite element analysis”, IOSR journal of Mechanical & Civil Engineering, ISSN: 2278-1684, PP 26-31

[5]. Shanmugasundaram Sankar, Maasanamuthu Sunder Raj, Muthusamy Nataraj, “Profile modification for increasing the tooth strength in spur gear using CAD ”, Scientific research Engineering, 2010, 2, 740-749

[6]. M. Rameshkumar, P. Shivkumar, S. Sundaresh & K. Gopiath, “Load sharing analysis of high contact ratio spur gears in military tracked vehicle applications”, gear technology, July 2010, PP 43-50

[7]. K. Mao, “Gear tooth contact analysis and its application in the reduction of fatigue wear”, Wear 262 (2007), PP 1281–1288

[8]. Babita Vishwakarma, Upendra Kumar Joshi, “Finite Element Analysis of Helical Gear Using Three-Dimensional Cad Model”, international journal of engineering sciences & research technology, April 2014, ISSN: 2277-9655

[9]. Amol Deshpande, Abhay Utpat, "Comparative analysis for bending and contact stresses of Girth Gear by using AGMA standard & finite element analysis", international journal of innovative research in science, engineering and technology, Vol. 3, Issue 10, October 2014, ISSN: 2319-8753

[10]. David H. Johnson, "Principle of simulating contact between parts using ANSYS"

[11]. D. SasiKanth, Tippa Bhimasankara Rao, "Design, Modeling and Analysis of Involute Spur Gears by Finite Element Method", International Journal of Engineering Research and Development, Volume 6, Issue 2 (March 2013), PP. 74-80, e-ISSN: 2278-067X

[12]. Seney Sirichai, Ian Howard, Laurie Morgan & Kian Teh, "Finite element analysis of gears in mesh", Fifth international congress on sound & vibration, Dec. 15-18, 1997

[13]. Mrs. C.M. Meenakshi, Akash Kumar, Apoorva Priyadarshi, Digant Kumar Dash and Hare Krishna, "Analysis of Spur Gear Using Finite Element Analysis", Middle-East Journal of Scientific Research 12 (12): 1672-1674, 2012, ISSN: 1990-9233

[14]. Ali Hammoudi, Amal A. Abdullah, "Non-Linear Analysis Spur Gear Mesh by Finite Element Method", Journal of Kerbala University , Vol. 5 No.4 Scientific, Dec. 2007

[15]. Raghava Krishna Sameer.B, V.Srikanth, "Contact stress analysis of modified helical gear using Catia and Ansys", international journal of Computer science information and Engineering Technologies ISSN: 2277-4408