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

Parag Vyankatrao Thote
Yashoda Technical Campus, Satara, 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.

Index Terms – FEA, Static & Dynamic Stresses, Starter Motor Pinion, Ring Gear

1. 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]. This paper presents the stress analysis of mating tooth of spur gears to find maximum contact stress in the gear tooth. The results obtained from Finite Element Method are compared with theoretical Hertzian equation values. The spur gears are sketched, modelled & assembled in ANSYS 14.5 Design Modular. The results show that the difference between maximum contact stresses obtained from Hertz equation & Finite Element Analysis is very less and it is acceptable also the deformation patterns of steel & cast iron gears depict that the difference in their deformation is negligible [3].

As mentioned in the paper a pair of spur gear tooth in action is generally subjected to two types of cyclic stresses: bending stresses inducing bending fatigue & contact stress causing contact fatigue. Both this type of stresses may not attend their maximum values at same point of contact fatigue. These types of failures can be minimized by careful analysis of the problem during the design stage & creating proper tooth surface profile, in order to analyze spur gear pair a 3D deformable-body (model) of spur gear is developed and bending stress analysis will be performed as it affects transmission [4]. In this paper tooth failure of spur gear is examined. The spur gear with less than 17 numbers of tooth had the problem of undercutting during gear manufacturing process which minimizes the strength of gear at root. Circular root filled instead of the standard trochoidal root filled is introduced in spur gear & analyzed using ANSYS. From the study it is observed that the circular root filled design is particularly suitable for lesser number of tooth in pinion &where as the trochoidal root filled design is suitable for higher number of tooth [5]. In this paper**,** the actual shape of trochoid is considered, whereas the fillet radius is assumed as a constant radius curve for calculating the geometry factor presented by AGMA. The FEA of final drive gear assembly of Military vehicle for both NCR & HCR is examined and the contact stresses are more than 25% less in HCR gears compared To NCR gears. The load carrying capacity of HCR gearing could be increased by at least 25% for the same weight & volume [6]. The present paper concentrates on the gear fatigue wear reduction through micro-geometry modification method. An advanced non-linear finite element method has been successfully used to accurately simulate gear contact behaviour. The models have used true three dimensional gear tooth profiles with micro-geometry modifications under real load conditions. The shaft misalignment, deflection and assembly deflection effects on gear surface contact behaviour have been investigated. The optimized micro-geometry based on the analysis has been proposed to reduce surface contact fatigue failure. The model has been very successfully applied in automotive transmission gear surface fatigue wear reduction. The highly accurate gear micro-geometry modification method has improved the gear surface fatigue wear significantly. This method can also be applied to transmission system noise analysis in term of transmission error reduction [7].

This paper investigates finite element model for monitoring the stresses induced of tooth flank, tooth fillet during meshing of gears. The involute profile of helical gear has been modelled and the simulation is carried out for the bending and contact stresses and the same have been estimated. To estimate bending and contact stresses, 3D models are generated by modelling software CATIA V5 and simulation is done by finite element software package ANSYS 14.0. Analytical method of calculating gear bending stresses uses Lewis and AGMA bending equation. For contact stresses Hertz and AGMA contact equation are used. Study is conducted by varying the face width to find its effect on the bending stress of helical gear. It is therefore observed that the maximum bending stress decreases with increasing face. In this work we got on three results as follow; Theoretical results (from Lewis equation and Hertz equation directly), AGMA results, ANSYS results and all results are closer [8]. The present work is aim of analytical design of girth gear by using AGMA standard; bending & contact stresses on the gear tooth are calculated. The validation of the stresses is done by using FEA (ANSYS). The values of the bending stress and contact stress determined using AGMA found to be in agreement with ANSYS results and corresponding error observed is less than 5%. Girth gears are large ring gears which are normally fitted to the outside of the mills or kilns to provide the primary rotational drive. The two things that form the test of the gear design are surface durability (pitting) and tooth bending strength. A girth gear can be single or double pinion driven [9]. The major objective of this project is to find the region of the surface distortion due to wear at vicinity of the pitch line. Generally for medium power transmission spur gears with involute profile is used. In transmitting the power, the gears are subjected to number of stresses and the failure of the gear is mainly caused due to bending and pitting. The former can be avoided by providing high strength material i.e. material having high static strength but the pitting failure can be avoided only by proper surface hardening of the gear teeth. Most of the gear failures occur mainly due to contact failures. Pitting occurs at the pitch point on the surface of the tooth. The stresses developed are called as Contact stresses. One more observation during this work is that σx varies distinctively about pitch line as if it is junction between two stress regions [10].

2. 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.

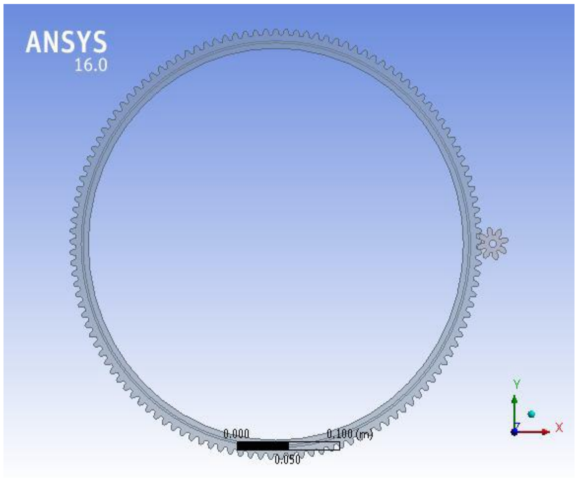

Figure1. Ring Gear and Pinion

- **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 1 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.

3. 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.

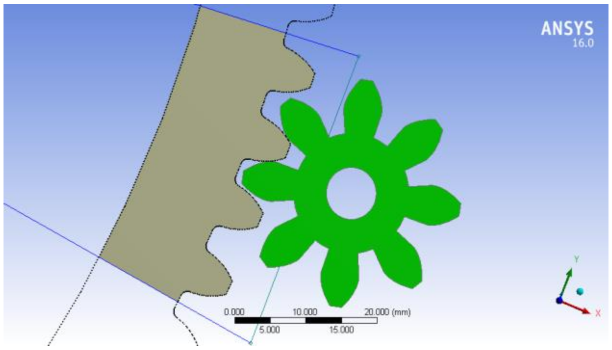

Figure 2 Working Geometry

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.

Table 2 Properties of Material

Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1	
Compressive Yield Strength	2.5e+008 (Pa)
Tensile Yield Strength	2.5e+008 (Pa)
Tensile Ultimate Strength	4.6e+008 (Pa)
Young's Modulus	2.e+011
Poisson's Ratio	0.3
Density	7850 kg m^-3
Reference Temperature	22 ^{0}C

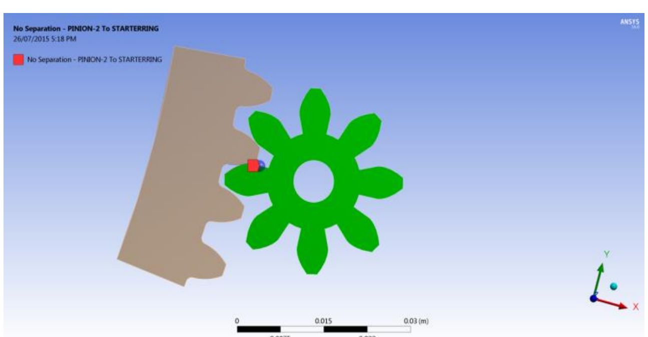

Figure 3 Contact between to gears

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

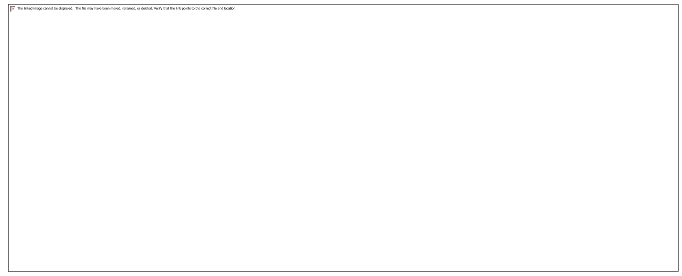

Figure 4 Fix support and moment

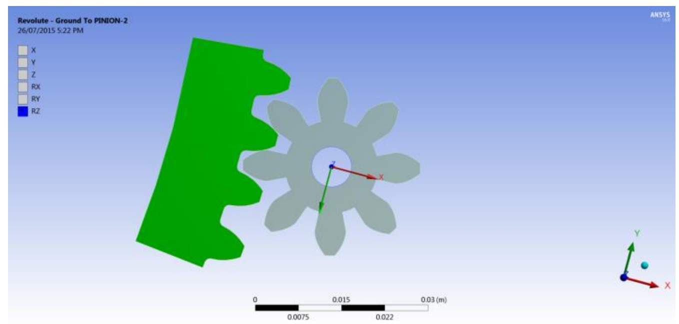

Figure 5 Direction of rotation of Pinion

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.

.

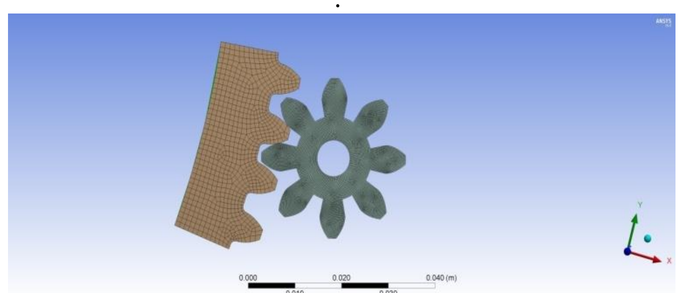

Figure 6 Meshing of geometry

Fine meshing is applied to pinion for obtaining more precise results i.e. contours after meshing number of nodes & elements are 11221 and 10744 respectively, meshing of the geometry shown in figure above;

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)

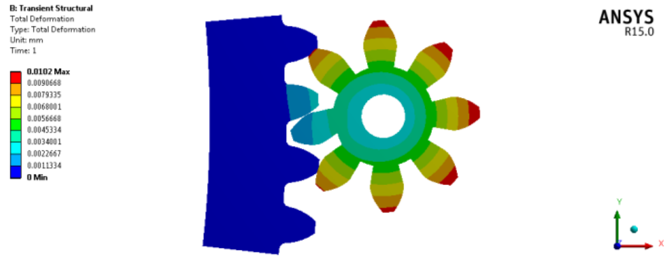

Figure 7 Deflection for working geometry

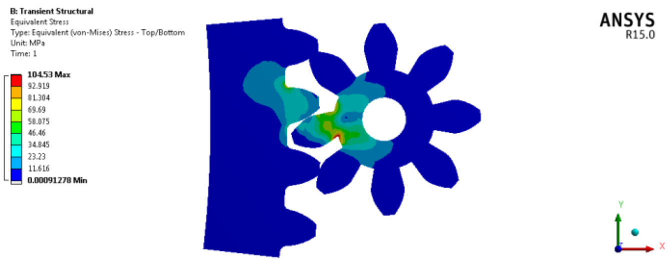

Figure 8 Equivalent stresses for working geometry

- **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)

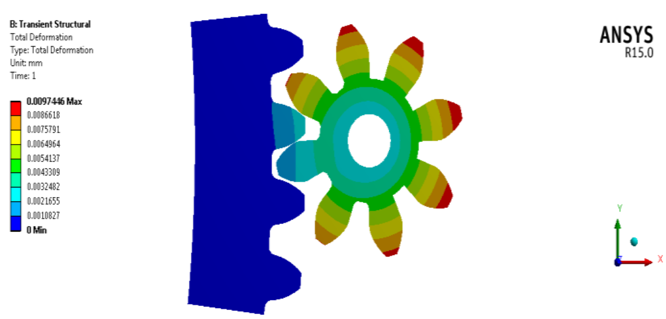

Figure 9 Deformation for root fillet 0.5mm

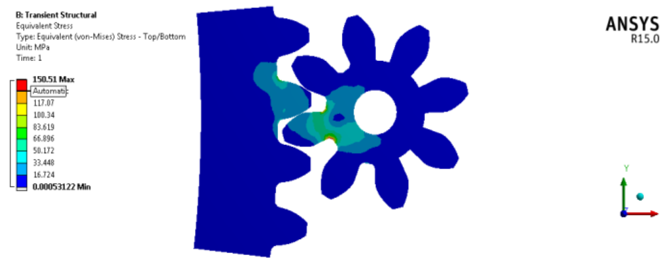

Figure 10 Equivalent stresses for root fillet 0.5mm

- **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)

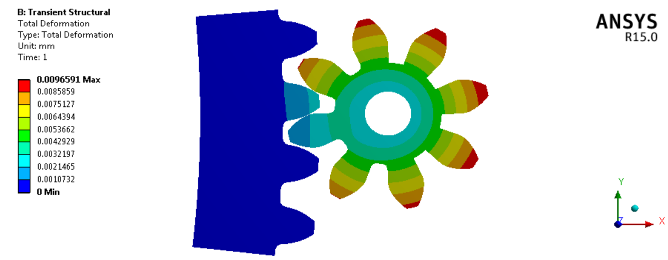

Figure 11 Deformation for root fillet 0.75mm

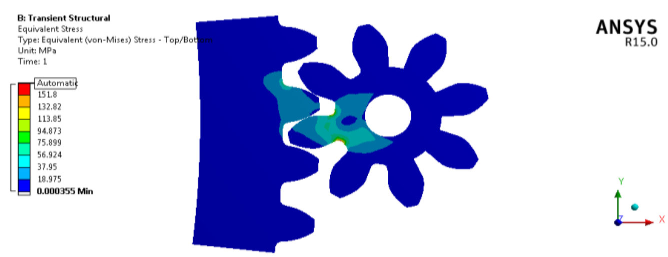

Figure 12 Equivalent stress for root fillet 0.75mm

- **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.

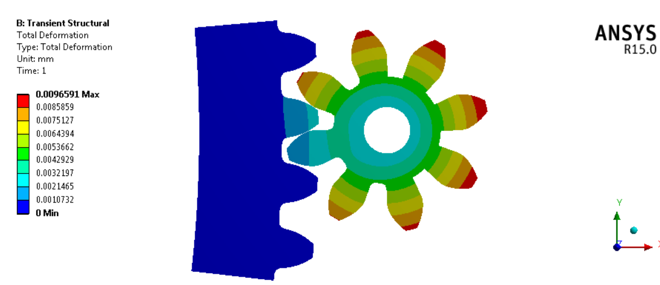

Figure 13 Deformation for root fillet 1mm

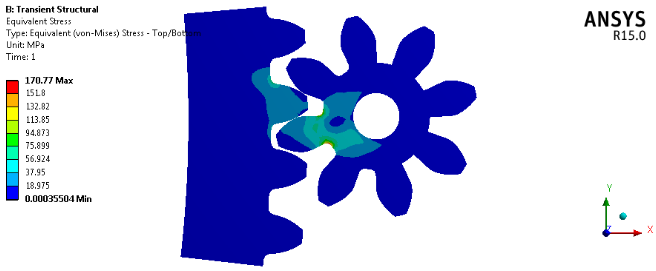

Figure 14 Equivalent stresses for root fillet 1mm

- **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.

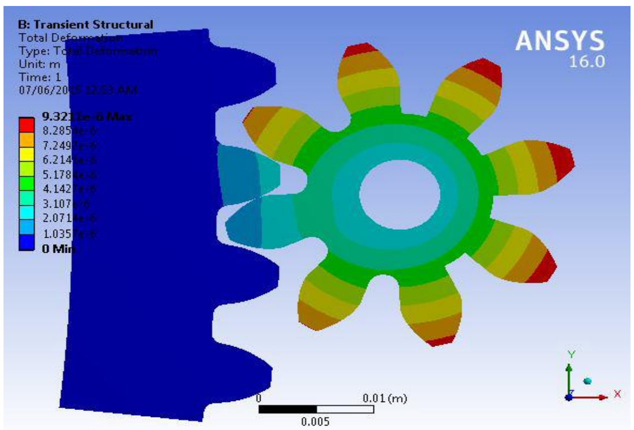

Figure 15 Deformation for root fillet 1.25mm

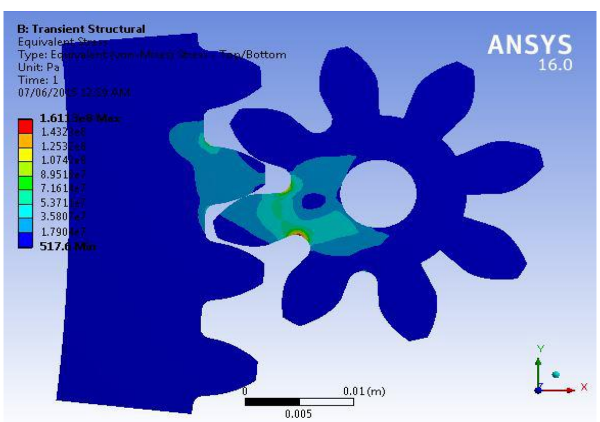

Figure 16 Equivalent stresses for root fillet 1.25mm

- **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.

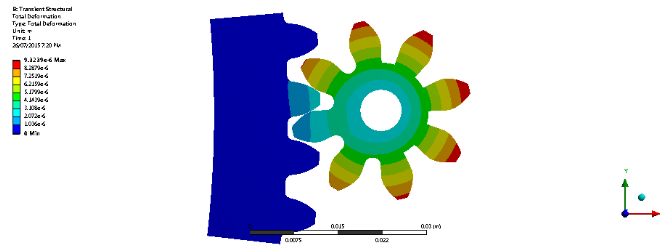

Figure 17 Deformation for root fillet 1.5mm

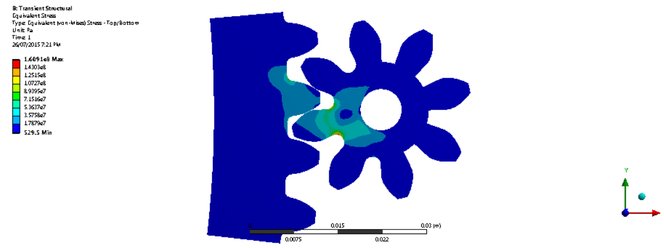

Figure 18 Equivalent stress for root fillet 1.5mm

4. 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.

The results of stresses obtained from photoelastic analysis are in good agreement with the results obtained from finite technique. At this stage of project completion it can be concluded that Finite Element method is very effective in order to predict stresses & deformations in system. Both the analysis techniques used in this study are non- destructive type, hence these techniques are very cost effective also. The error observe while comparing results can be because of involvement of human being. The observation capacity & excellent operating skill is required by an operator while using polariscope, even a small variation in observation can lead to high fluctuation in final results.

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

Author

Mr. Parag Vyankatrao Thote is working as an Assistant Professor in Department of Mechanical Engineering at Yashoda Technical Campus, Satara. He has completed M.E. Mechanical (Automotive) from SPPU, Pune. The author has published several papers in various national and international journals. His areas of interests are wind energy, Autotronics, waste heat recovery, Biomass waste utilization and Mechatronics.