Investigation of Bending and Contact Stresses of Helical Gear Pair using Finite Element Method and Validated by ISO 6336

Prof. Digvijay Gajanan Bhosale
Assistant Professor,
Department of Automobile Engineering,
Sanjeevan Engineering and Technology Institute,
Panhala, Kolhapur- 416 201,India.

Abstract- **The traditional gear stress analysis technique is based on the International (ISO 6336:2006) and American (ANSI/AGMA) are empirically based comparator approaches for establishing contact and bending stresses in spur and helical gears. A limitation of both ISO and AGMA is a failure to correctly account for benefits from using optimized micro gear geometry accounting for the systematic deflections and manufacturing errors that increases static and dynamic stresses. In addition, they do not account for geometrical changes due to centrifugal loading and thermal expansions nor do they account for externally generated vibrations. An alternative stress analysis technique using a commercially available Finite Element Analysis Program ANSYS 12.1 is chosen to investigate the feasibility of using FEA for optimized gear stress analysis. Coupled with high performing computing technology, ANSYS is symmetrically validated against the respective ISO 6336:2006. So using finite element methods it is possible to carry out bending stress and contact stress analysis of helical gear.**

Keywords— Dynamic stresses, helical gear ,Optimised gear stress analysis

I. INTRODUCTION

Designing highly loaded helical 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. 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 modelling time, a pre-processor method that creates the geometry needed for a finite element analysis may be used, such as that provided by CAD/ Engineer. CAD/Engineer can generate models of three-dimensional gears easily. In CAD software, the geometry is saved as a file and then it can be transferred from it to ANSYS.

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 gear box under consideration is manufactured by Ghatage Patil Industries Pvt Ltd. and is primarily used for fishing travelers small tags wherein the vessel is used, more or less at maximum flywheel torque condition without much variation in the engine load conditions and at constant cruising speed. Specifications of Gear Box are as follows:

- Maximum Torque : 745.56 N.m
- Maximum Input RPM : 2000
- Reduction Ratio : 3.94:1
- Input Rotation : Anti-cloclwise
- Output Rotation : Clockwise or Anti-cloclwise depending upon actuating lever position
- Maximum Power Rating : 156 kW at 2000 RPM

II. TOOTH ROOT BENDING STRENGTH FOR PINION AND GEAR USING ANSYS

In this section the tooth root stresses and the tooth deflection of one tooth of a helical gear is calculated using an ANSYS model.

A. Build Geometry

Construct a three dimensional representation of the pinion tooth to be modelled and tested using the work plane coordinates system within ANSYS. Model of the pinion tooth can be build or created in any of the CAD software[9] and then it is import into ANSYS for analysis.

B. Meshing

The element type[7-8] "SOLID TETRAHEDRAL 10 NODES 187" was chosen.. SOLID187[2] element is a higher order 3-D, 10-node element. SOLID187 has a quadratic displacement behaviour and is well suited to modelling irregular meshes (such as those produced from various CAD/CAM systems). The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, creep, stress stiffening, large deflection, and large strain capabilities.

C. Applying constraints

While analyzing or solving problem of pinion, i.e. to determine bending stresses induced in gear tooth it is required to use" LEWIS EQUATION" & we know that in

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 07, July-2016

Lewis equation, the gear tooth is treated as "CANTILEVER BEAM". The tangential load acting on gear tooth causes the bending stresses at the base of the tooth[2]. Following figure shows how gear tooth is constrained in order to determine bending stresses in finite element analysis software.

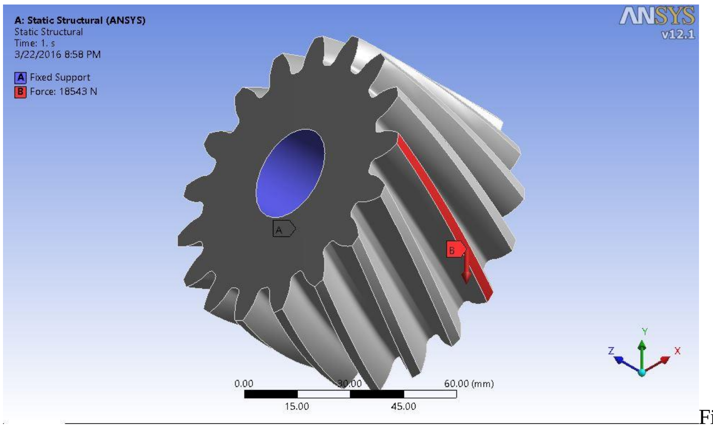

Fig 1. Pinion tooth with constraints

III. RESULTS USING ANSYS ANALYSES

For pinion tooth root stress

From figure 2 of stresses in Y-Direction ,

σ_f = Tooth root bending strength for pinion = 346.87 N/mm^2,

σ_{FP} = Permissible tooth root bending stress for pinion=911 N/mm^2

We have, $\sigma_{FP} \geq \sigma_f$

& σ_{FP} =911/ SF

$\therefore$ 911/ SF $\geq$ 346.87

$\therefore$ 911/ SF = 346.87

SF = 2.62, safety factor

The recommended SF for root stress for main propulsion gears, shall be in the range of 1.55 & 3.5

For gear tooth root stress

σ_f = Tooth root bending strength for pinion = 223.32N/mm^2,

σ_{FP}= Permissible tooth root bending stress for pinion=942.88 N/mm^2

We have, $\sigma_{FP} \geq \sigma_f$

& σ_{FP} =911/ SF

$\therefore$ 911/ SF $\geq$ 217.54

$\therefore$ 911/ SF = 217.54

SF = 4.07, safety factor

The recommended SF for root stress for main propulsion gears, shall be in the range of 1.55 & 3.5

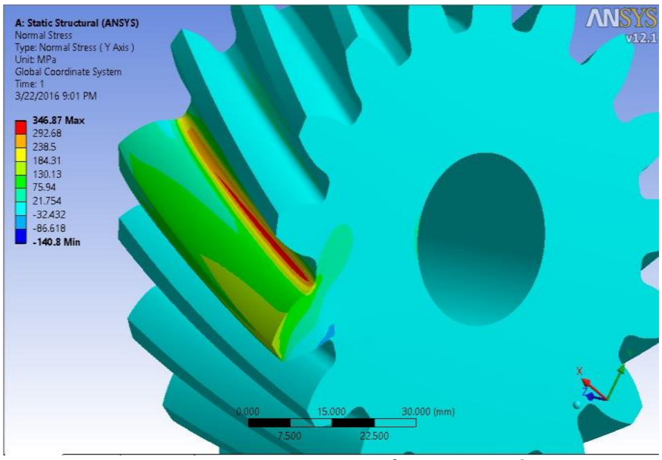

Fig. 2. Stresses in Y-Direction for Pinion Tooth

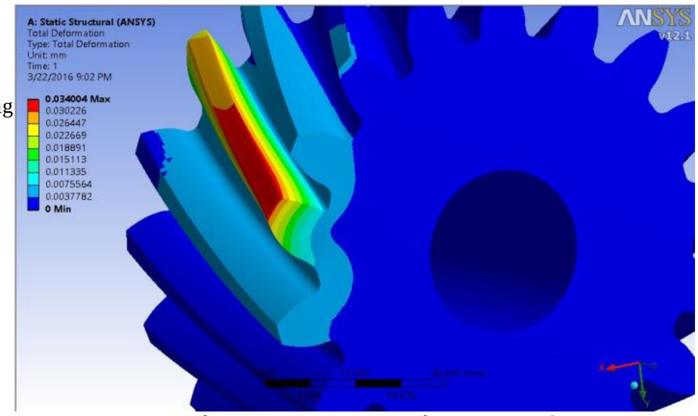

Fig. 3. Deformation in Y-Direction for Pinion Tooth

IV. LOAD CARRYING CAPACITY OF GEAR PAIR

- Bending Stress(σ_F) - Ref. Section 3, of ISO 6336 / 3[1]

The root stress σ_F must not exceed permissible root stress σ_{FP} for pinion as well as for gear. The root stress σ_F is given by,

$$\sigma_F = \frac{F_t}{b.m_n} \quad y_f . y_s . y_\beta . k_A . k_v . k_{F\beta} \qquad (1)$$

By considering all of these factors the tooth root stress for pinion, σ_F = 381.261 N / mm^2 and that for gear σ_F = 444.972 N / mm^2

Permissible root stress is given by, [1-6]

$$\sigma_{FP} = \sigma_{FE} . y_d . y_n \; . \; y_{\delta relT} . \; y_{RrelT} . \; y_x \qquad (2)$$

By considering all of these factors the permissible tooth root stress for pinion,

σ_{FP} = 911 N / mm^2 and that for gear σ_{FP} = 942.885 N / mm^2

- Contact stress (σ_H) - It can be determined by , [1-4]

$$\sigma_H = Z_H . Z_E . Z_\varepsilon . Z_\beta \sqrt{\frac{F_t}{b.d_1}} \sqrt{\frac{u+1}{u}} \qquad (3)$$

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 07, July-2016

By considering all above factors contact stress will be, σ_H = 977.683 N / mm ^{2}

Determination of permissible contact stress σ_{HP},[1-5]

$$\sigma_{HP} = \sigma_{H\,lim}.Z_{NT}\, Z_L\, Z_V\, Z_R\, Z_W\, Z_X \qquad (4)$$

- $\sigma_{H\ lim}$ = the allowable stress no (contact) which accounts for the influence of the material ,heat treatment and surface roughness for standard reference test gear
- Z_{NT} = the life factor for test gears for contact stress which accounts for higher load carrying capacity for limited no. of load cycles.
- Z_L = the lubricant factor which accounts the lubricant viscosity
- Z_R = the roughness factor which accounts for the surface roughness
- Z_W = the work hardening factor which accounts for the effect of meshing with a surface hardened or similarly hard mating gear
- Z_V = the velocity factor which accounts for influence of pitch line velocity
- Z_X = the size factor for contact stress which accounts for the influence of the tooth dimensions for the permissible contact stress

σ_{HP} = 2368.4929 N / mm ^{2}

V. CONTACT ANALYSIS USING ANSYS

In this section the tooth contact stresses and the tooth deflection of a helical gear is calculated using an ANSYS model. For the contact stresses, the numerical results are compared with the values given by the draft proposal of the standards as calculated in previous section. Despite the importance of contact in the mechanics of solids and its engineering applications, contact effects are rarely seriously taken into account in conventional engineering analysis, because of the extreme complexity involved[3]. Usually the loading causes significant changes in stiffness, which results in a structure that is nonlinear. Nonlinear structural behavior arises for a number of reasons, which can be reduced to three main category[3]

1.Geometric Nonlinearities (Large Strains, Large Deflections)
2.Material Nonlinearities (Plasticity)
3. Change in Status Nonlinearities (Contact).

In order to handle contact problems in meshing gears with the finite element method, the stiffness relationship between the two contact areas is usually established through a spring that is placed between the two contacting areas. This can be achieved by inserting a contact element placed in between the two areas where contact occurs. There are many types of contact problems that may be encountered, including contact stress, dynamic impacts, metal forming, bolted joints, crash dynamics, assemblies of components with interference fits, etc. All of these contact problems, as well as other types of contact analysis, can be split into two general classes[3]
1.Rigid - to - flexible bodies in contact,
2.Flexible - to - flexible bodies in contact

In general, there are three basic types of contact modelling application as far as ANSYS use is concerned[2-3].
1. Point-to-point contact: the exact location of contact should be known beforehand. These types of contact problems usually only allow small amounts of relative sliding deformation between contact surfaces.
2. Point-to-surface contact: the exact location of the contacting area may not be known beforehand. These types of contact problems allow large amounts of deformation and relative sliding. Also, opposing meshes do not have to have the same discretisation or a compatible mesh.
3. Surface-to-surface contact is typically used to model surface-to-surface contact applications of the rigid-to-flexible classification

A. Build geometry

Construct a three dimensional representation of the gear to be modelled and tested using the work plane coordinates system within ANSYS. Model of the pinion and gear pair can be build or created in any of the CAD software[9] i.e. CATIA V5 and then it is converted into STP format before importing into ANSYS for analysis.

B. Applying constraints

While analyzing or solving problem of gear pair in contact, i.e. to determine contact stresses induced in gear tooth it is required to constraint the geometry properly so that there is no weak spring error or rigid body displacement error which is due to under constrained geometry. This type of error mostly occurs during non linear structural analysis. Following figure shows how gear pair is constrained in order to determine contact stresses in finite element analysis software. There are three boundary conditions namely[2]
1. Fixed support- applied to bottom surface of gear sector which will freeze all DOF of gear.
2. Cylindrical support- applied to hole surface of gear in which radial and axial movements of pinion are fixed and only tangential movement will be free which will allow pinion to rotate about its own axis
3. Torque- applied on the pinion in seven load steps which are as shown in the graph below

No of iteration required by the solver to converge each of the applied load step is shown in the graph below. Since in nonlinear problems initial seed means initial guess value will be determined by program itself some iteration are required to be performed. If initial guess value is within the radius of convergence then solution will converge quickly and desired results will obtained. If initial guess is outside of radius of convergence then it is very difficult to converge nonlinear solution.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 07, July-2016

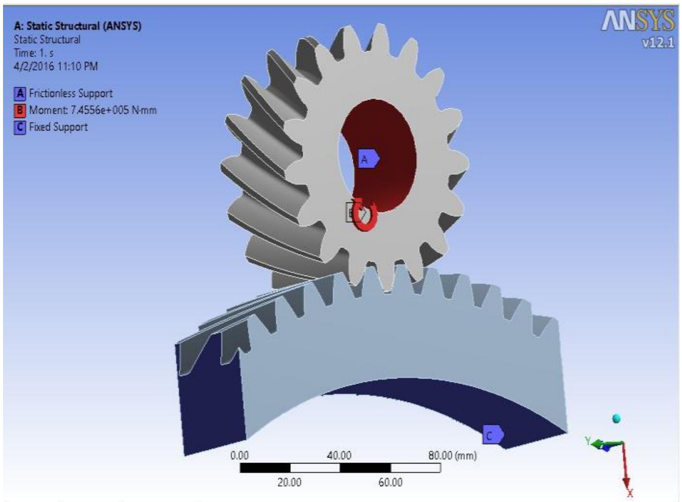

Fig. 4. Gear Pair Model With Boundary Conditions

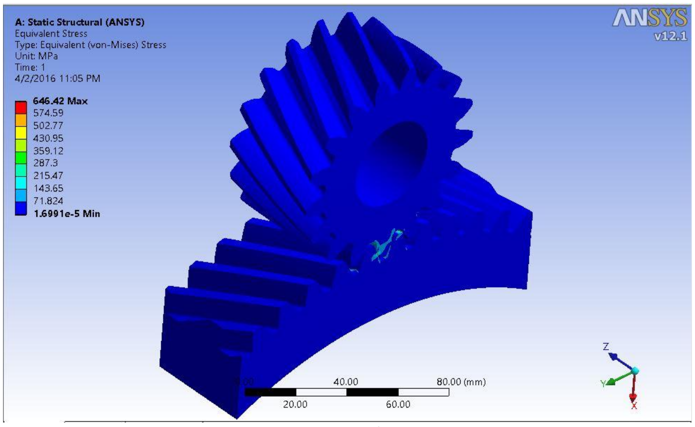

Fig. 5.Contact Stresses for Gear pair

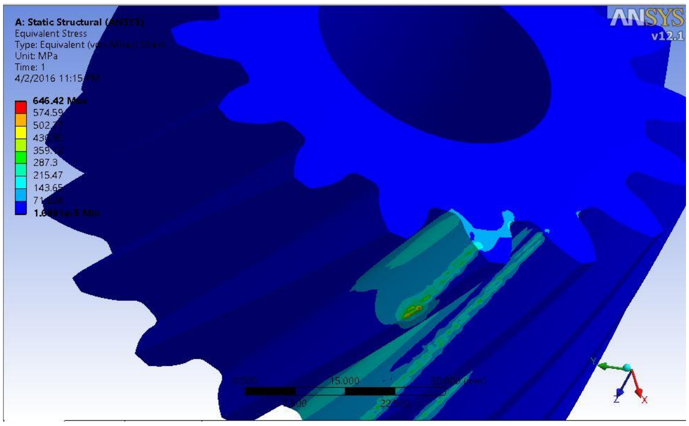

Fig. 6.Contact Stresses for pinion

VI. CONCLUSION

Bending Analysis: This analysis is based on the conventional design of gear in which the gear tooth is considered as cantilever beam. From this analysis it is found that the gear and pinion teeth are very much on safer side for flexural or bending strength of the material is having much higher value.

Contact Analysis: The maximum von mises stress should occur at the contact point between the two meshing teeth of gears. The von mises criterion is best applied when used to predict the onset of yielding in a structure where the material behaves in a ductile fashion. From analysis it is clear that, contact stresses induced are much below the permissible contact stresses of gear material.

So it is possible to optimize the gear material which will cost effective as well for organization with respect to raw material cost and manufacturing cost.

REFERENCES

[1] Shuting Li , *"Finite element analyses for contact strength and bending strength of a pair of spur gears with machining errors, assembly errors and tooth modifications"*, Science Direct/ Mechanism and Machine Theory 42 (2007) 88–114.

[2] Chien-Hsing Li , Hong-Shun Chiou , Chinghua Hung, Yun-Yuan Chang , Cheng-Chung Yen, *"Integration of finite element analysis and optimum design on gear system"*, Elseveir / Finite Elements in Analysis and Design 38 (2002) 179-192.

[3] Yi-Cheng Chen, Chung-Biau Tsay , *"Stress analysis of a helical gear set with localized bearing contact"*, Elseveir / Finite Elements in Analysis and Design 38 (2002) 707–723.

[4] Kuo Jao Huang , Hsin Wei Su, *"Approaches to parametric element constructions and dynamic analyses of spur/helical gears including modifications and undercutting"*, Elseveir / Finite Elements in Analysis and Design 46 (2010) 1106–1113.

[5] Load carrying capacity of Spur and Helical gears , 2006. ISO 6336/2006.

[6] ANSYS12.1 Manual, 2009. ANSYS Inc.

[7] Zeping Wei , *Stresses and deformations in involute spur gears by finite element method*.

[8] G. M. Maitra and L. V. Prasad , Reprint 2000. PSG Design Data , PSG College of Technology , Coimbatore .

[9] Second Edition. Handbook of Mechanical Design , Tata McGraw Hill Publishing Company Ltd..

[10] Joseph E. Shigley and C. R. Mischke , Sixth Edition. Mechanical Engineering Design, Tata McGraw Hill Publishing Company Ltd.

[11] J. N. Reddy. An Introduction to Finite Element Method, McGraw Hill International Edition.

[12] Tirupathi R. Chandrupatla and Ashok D. Belegundu ,Third Edition. Introduction to Finite Elements in Engineering, Pearson Publication .

[13] N. D. Bhatt and V. M. Panchal ,Thirty Nineth Edition. Machine Drawing ,Charotar Publishing.