FEA of Polyethylene Helical Gear Set

[1]Vinaykumar C. Jadhav, [2]Kiran B. Gore, [3]Dinesh S. Suryavanshi
[1]Assistant Professor, [2]Assistant Professor, [3]Assistant Professor
[1]Department of Mechanical Engineering,
[1]S.N.J.B.'s Late K.B.J. College of Engineering, Chandwad, Nashik, India.

Abstract: The main focusing areas of this analysis are to Modelling the gear without losing its geometry in Pro/engineer software. To analyze and compare the various stresses, strain and safety factors of the model of helical gears made of Polyethylene alloy using FEM tool ANSYS. Generate the profile of helical gear teeth model to calculate the effect of gear bending using three-dimensional model and compare the results. Performing parametric study to study the effect of varying load on the various stresses, strains on the helical gear model.

Index Terms **- Helical Gear, FEM, Polyethylene, stress, strain, directional deformation, gear bending.**

I. INTRODUCTION

There have been so many research works by so many researchers in the system of gear. Still there is an approach, which is able to determine the deviation in the geometry of helical gear like contact and bending stresses.[1] The main focusing area in helical gears is the dynamic load, noise level during operation, weight, and size that should be lighter and small accordingly.[2] Such type of gears fails at the root of the teeth because of lower bending strength and due to the action of pitting which can be controlled by implementing the correct method for analysis and the modification of the different gear geometry parameters. [3]In view of this approach, the main objective of this analysis is to develop model of helical gear set in meshing and to evaluate the effect of various stresses on gear tooth. The main objective of this analysis is as follow:

1.1 Modelling the 3 D model of gear set in Pro/engineer workbench.
1.2 Evaluation and analysis of the various stresses, strain and factors of safety of helical gear set made of Polyethylene using FEM tool ANSYS.
1.3 Generate the helical gear set model to calculate the effect of gear bending in teeth,
1.4 To study the effect of varying load ranging from 1 KN to 10 KN on the various stresses, strains on the Helical gear set.

II. SOLID MODELING AND FEM PAKAGES

2.1 Geometrical Parameter and Material Properties

For generating the solution here ANSYS is used to perform the analysis. Steps used for obtaining the solution using ANSYS 14.0

1. The geometry of the gear is imported from 3D modeling workbench Pro/Engineer, in IGES format.
2. The element type and materials properties are specified.
3. Next is Meshing.
4. The boundary conditions are applied.
5. The solution is obtained on the basis of input parameters.
6. Finally, the solution is obtained in which is evaluate to obtained results.

Table.1 Key Specifications of Geometrical Parameters of the Polyethylene Helical Gear

Sr.No	Geometry Name	Gear 1	Gear 2
1	No. of teeth	20	32
2	RPM	10000rpm	6250 rpm
3	Rotational velocity	1047 rad/sec	655rad/sec
4	Diameter pitch	200 mm	320 mm
5	Addendum	220 mm	340 mm
6	Dedendum	180 mm	278 mm
7	Face width	6.67 mm	7.104 mm
8	Material type	Polyethylene	Polyethylene

2.2 Material Properties

Table 2.ASME, Section II, Part D Material Properties

Sr.No.	Polyethylene Material Properties	
1	Young's Modulus of Elasticity (Gpa) E	0.8 Gpa
2	Ultimate Tensile Strength	15 MPA
3	Poisson's Ration (U)	0.46

III. FINITE ELEMENT FORMULATION

For the representation of element here three-dimensional eight-nodded solid element is chosen which is from hexahedral family. The three dimensional eight nodded solid element as shown in Fig.1 has eight nodes which are located at each of the corners and has three translational degree of freedom for each of the node.

3.1 Modeling of Helical Gear

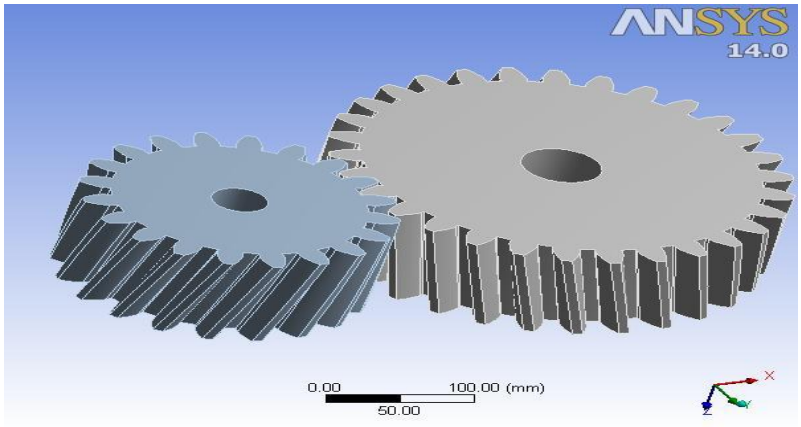


Fig.3.1 Imported model of Helical Gear set in ANSYS 14 from Pro/E

3.2 Boundary Conditions

As the wheel and pinion is in mesh the contact type analysis is selected in ANSYS. Contact type analysis is a Non- linear type analysis. Among the types of contact available in the ANSYS module, the rough contact is selected as power is transmitted due to friction between wheel and pinion. As the contact is rough between wheel and pinion the contact surface behaves like spring for such types of contact 'Pure Penalty Method' is chosen from ANSYS module.

3.3 Meshing

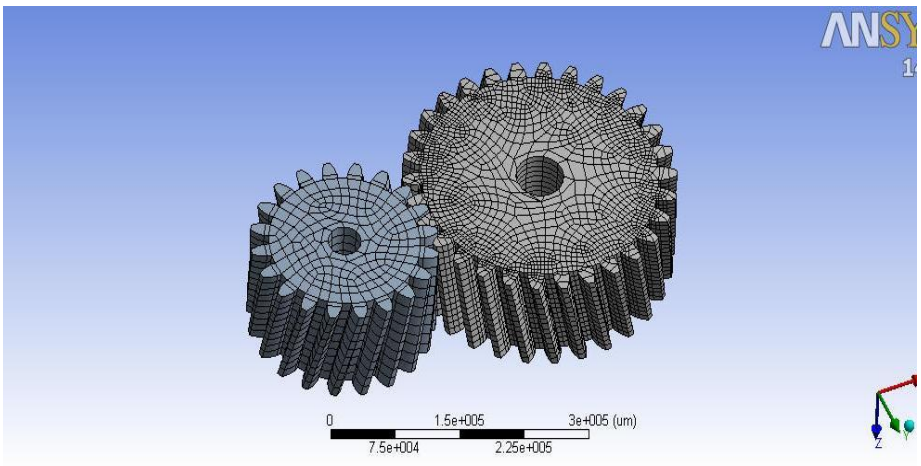


Fig.3.2 Mesh 3D Model of Helical Gear Set

IV. ANALYSIS AND RESULT

4.1 Deformation, Stress, Strain, Safety Factor Analysis

Table.4.1 Analysis Result of Polyethylene Helical Gear Set

Name of analysis	Polyethylene Helical gear	
	Maximum	Minimum
Total Deformation	1.9108 mm	0.99262 mm
Directional Deformation	0.60267 mm	-0.17226 mm
Equivalent Elastic Strain	2.1307e-002 mm/mm	1.3621e-006 mm/mm
Shear Elastic Strain	2.372e-002 mm/mm	-2.7873e-002 mm/mm
Equivalent Stress	22.398 MPa	1.2997e-003 MPa
Shear Stress	9.1871 MPa	-10.796 MPa
Strain Energy	98.734 mJ	6.2816e-006 mJ
Stress Intensity	25.036 MPa	1.4991e-003 MPa
Safety factor	---	1.1162

4.2 Total Deformation at Varying Load Ranging from 1KN to 10KN

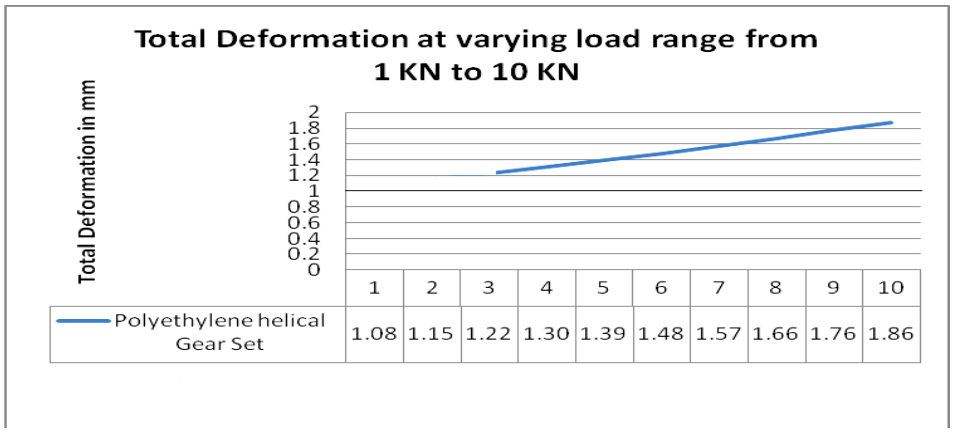


Fig.4.1 Graphical Representation of Total Deformation

4.3 Directional Deformation at Varying Load Ranging from 1KN to 10KN

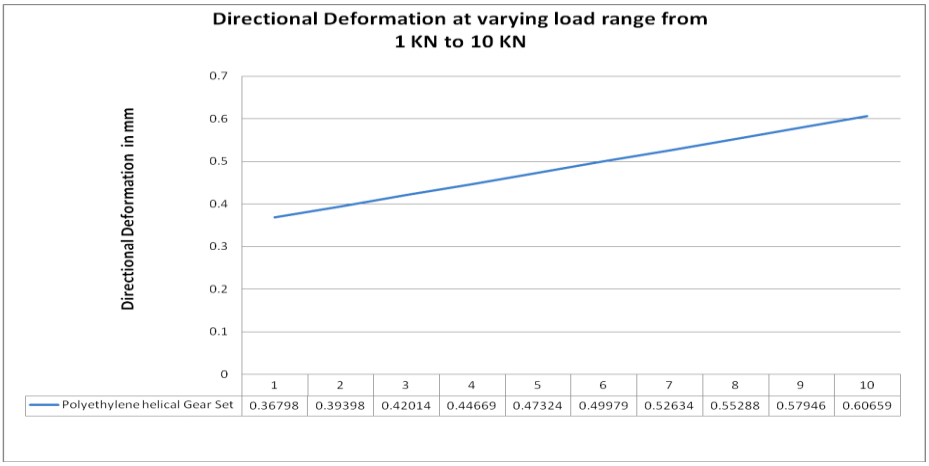


Fig.4.2 Graphical Representation of Directional Deformation

4.4 Equivalent Von-Mises Stresses at Varying Load Ranging from 1KN to 10KN

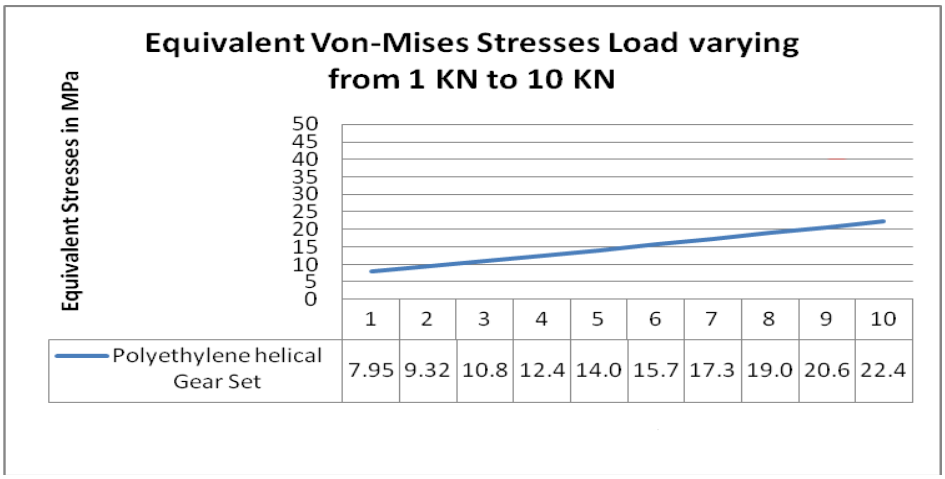
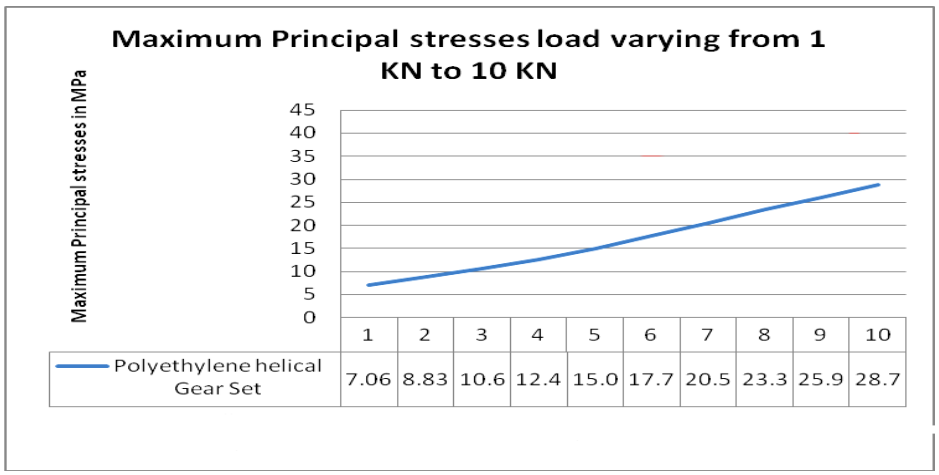


Fig.4.3 Graphical Representation of Equivalent Von-Mises Stresses

4.5 Maximum Principle Stresses at Varying Load Ranging from 1KN to 10KN



CONCLUSION AND FUTURE WORK

1. At varying load the stresses developed in a Polyethylene gear set are higher.
2. In case of various strain same thing observed.
3. Failure by Pitting that takes place on surface of the gear tooth will be diminish by improving the material hardness and surface finish.
4. Maximum bending stress can be minimizing by increasing face width of gear tooth.
5. If material strength value is have the prime importance then a gear with larger face width of any required helix angle can select.

Analysis of bending and contact stresses for all types of gears by three dimensional numerical method can be possible. Same FEA analysis can be conducted on the whole gearbox to evaluate mode of failure. The bending and contact stress analysis is possible on the gears which are of composite materials by using three-dimensional FEA.

REFERENCES

[1]. Purkar Sanjay and Pathak Sunil , Aspect of Finite Element Analysis Methods for Prediction of Fatigue Crack Growth Rate, Res. J. Recent Sci, Vol. 1(2), 85-91, Feb. 2012

[2]. Purkar Sanjay, Analysis of Crack Initiation in Fretting Fatigue Specimen, ISCA J. Engineering Sci.,1(1),26-34 2012

[3]. Vinaykumar C. Jadhav, Rahulkumar M. Sonar, Santosh D. Sancheti,"FEA of Titanium Helical Gear Set, in proceeding of ICRTET 2016 published by McGraw Hill publication, ISBN No – 978-93-392-2288-8, April 28-30, 2016.

