CONTACT STRESS ANALISYS OF A TRI TANGENT FILLET HELICAL GEAR TOOTH

K. Jhansi Rani[1], M. Venkaiah[2]

[1]*M.Tech Student Narasaraopeta Engineering College, Guntur dt, Andhra Pradesh, India*

Email: jansiramesh14@gmail.com

[2]*Assistant Professor, Department of Mechanical Engineering, Narasaraopet Engineering College, Narasaraopet, Andhra Pradesh, India*

Abstract: **In this project to change the design of involute profile by using CATIA V5, here the Helical gear tooth profile is designed by using tri tangent fillet tool and it is followed by the ANSYS. Main reasons are bending stresses and vibrations for the failure of a Helical gear in power transmission systems. To minimize the mass percentage bending stresses and vibrations in gears, contact stress analysis modified of a helical gear tooth is to be done to formulate the results. The analysis is done by using ANSYS workbench by dumping the design from CATIAV5, analytical study is based on Hertz's equation. Results of contact stresses on both standard and modified helical gears are to be taken and compared. If the solution of the modified helical gear is acceptable, The same design is chosen as an optimum for the fabrication of a helical gear. Helical gears having the leading edges of the teeth are not parallel and with set at an angle. Sometimes they are called as dry fixed or skew gears according to the configuration, as they are used for high speed applications and large power transmission due to maximum tooth face contact and one of the disadvantage is a resultant stress along the axis of the gear.**

I. Introduction

Gears are most commonly used for power transmission in all the modern devices. These toothed wheels are used to change the speed or power between input and output. They have gained wide range of acceptance in all kinds of applications and have been used extensively in the high-speed marine engines. In the present era of sophisticated technology, gear design has evolved to a high degree of perfection. The design and manufacture of precision cut gears, made from materials of high strength, have made it possible to produce gears which are capable of transmitting extremely large loads at extremely high circumferential speeds with very little noise, vibration and other undesirable aspects of gear drives.

II. LITERATURE SURVEY

Yonatan et al

Did the estimation of the bending stress, three-dimensional solid models for different face width are generated by Pro/Engineer that is a powerful and modern solid modeling software and the numerical solution is done by ANSYS, which is a finite element analysis package. The analytical investigation is based on Lewis stress formula[1].

Gears are mainly used to transmit the power in mechanical power transmission systems .These gears play a most predominant role in many automobile and micro electro mechanical systems. One of the main reason of the failure in the gear is bending stresses and vibrations also to be taken into account. But the stresses are occurred due to the contact between two gears while power transmission process is started.

Vijaya rangan et al

To estimate the bending stress, three-dimensional solid models for different face width are generated by Pro/Engineer that is a powerful and modern solid modeling software and the numerical solution is done by ANSYS, which is a finite element analysis package[2].

.Rao, C.M., and Muthuveerappan G

In this paper an attempt has been made to study the contact stresses of a pair of mating gear teeth, under static conditions, by using a two-dimensional finite element method and the Lagrangian multiplier technique[3].

Singiresu s. Rao et al

Finite Element Analysis is an analytical engineering tool developed in the 1960's by the Aerospace and nuclear power industries to find usable, approximate solutions to problems with many complex variables. It is an extension of derivative and integral calculus, and uses very large matrix arrays and mesh diagrams to calculate stress points, movement of loads and forces, and other basic physical behaviours[4].

III. PROBLEM DESCRIPTION

Helical gears offer a refinement over spur gears. The leading edges of the teeth are not parallel to the axis of rotation, but are set at an angle. Since the gear is curved, this angling causes the tooth shape to be a segment of a helix. Helical gears can be meshed in a *parallel* or *crossed* orientations. The former refers to when the shafts are parallel to each other; this is the most common orientation. In the latter, the shafts are non-parallel. The angled teeth engage more gradually than do spur gear teeth causing them to run more smoothly and quietly. With parallel helical gears, each pair of teeth first make contact at a single point at one side of the gear wheel; a moving curve of contact then grows gradually across the tooth face to a maximum then recedes until the teeth break contact at a single point on the opposite side. Quite commonly helical gears are used with the helix angle of one having the negative of the helix angle of the other; such a pair might also be referred to as having a right-handed helix and a left-handed helix of equal angles. The two equal but opposite angles add to zero: the angle between shafts is zero – that is, the shafts are *parallel*. Where the sum or the difference (as described in the equations above) is not zero the shafts are *crossed*. For shafts *crossed* at right angles the helix angles are of the same hand because they must add to 90 degrees.

IV. RELATED WORK CATIAV5 (COMPUTER AIDED THREE DIMENSIONAL INTERACTIVE APPLICATION)

As the world's one of the supplier of software, specifically intended to support a totally Integrated product development process. Dassault Systems (DDS) in recognized as a strategic partner. Catia Mechanical design solution will improve our design productivity. Catia is a suit of programs that are used in design, analysis and manufacturing of a virually unlimited range of the product. As of this software some of the tools are used design the helical gear are PAD: which adds the material according to the profile created in the sketcher module CIRCULAR PATTERN: This option is used to rotate the teeth with respect to the gear axis which generates total number of teeth on complete crown of the gear TRITANGENT FILLET: This tool is used to create the fillet along the teeth, which creates tangential fillet. The following figures shows the standard gear and modified gear.

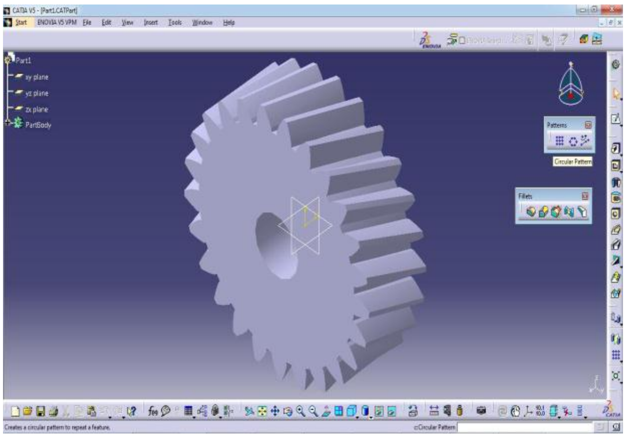

Fig.1 ISOMETRIC VIEW OF STANDARD HELICAL GEAR

V. MODULES IN CATIA

Sketcher module enables us to create sections. Sketcher technique is used in many areas of Catia. Using Sketcher mode, we can create geometry without regard to the exact relationships between parts of sketch or the cxact value of dimensions, when we generate the sections, Catia makes explicit assumptions. For example if we draw nearly horizontal line, it becomes exactly horizontal and all these assumptions are displayed graphically.

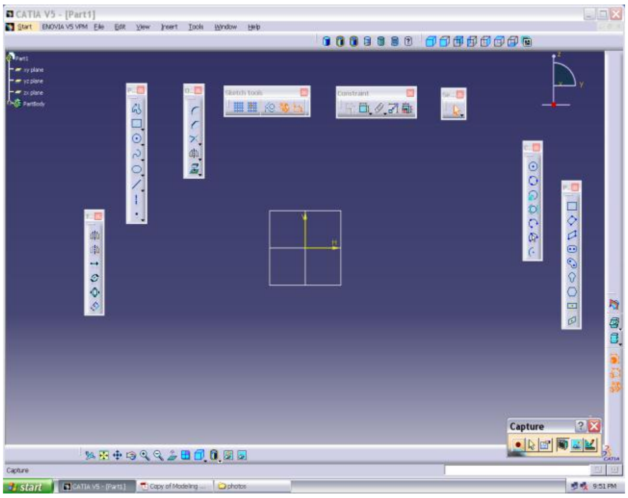

Fig.**2 SKETCHER MODULE**

VI. FINITE ELEMENT METHODS

The finite element method is numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering schools and industries. In more and more engineering situations today, we find that it is necessary to obtain approximate solutions to problems rather than exact closed form solution. It is not possible to obtain analytical mathematical solutions for many engineering problems. An analytical solutions is a mathematical expression that gives the values of the desired unknown quantity at any location in the body, as consequence it is valid for infinite number of location in the body. For problems involving complex material properties and boundary conditions, the engineer resorts to numerical methods that provides approximate, but acceptable solutions.

TABLE -1 HELICAM GEAR MODEL DEVELOP

Number of teeth	25
Diametric pitch (p) [mm]	60
Pressure angle	20 degree
Addendum [mm]	1/p
Dedendum [mm]	1.25/p
Helix angle	12 degree

A. General Description of FEM

In the finite element method, the actual continuum of body of matter like solid, liquid or gas is represented as an assemblage of sub divisions called Finite elements. These elements are considered to be inter connected at specified points known as nodes or nodal points. These nodes usually lie on the element boundaries where an adjacent element is considered to be connected. Since the actual variation of the field variables (like Displacement, stress, temperature, pressure and velocity) inside the continuum are is not know, we assume that the variation of the field variable inside a finite element can be approximated by a simple function. These approximating functions (also called interpolation models) are defined in terms of the values at the nodes. When the field equations (like equilibrium equations) for the whole continuum are written, the new unknown will be the nodal values of the field variable. By solving the field equations, which are generally in the form of the matrix equations, the nodal values of the field variables will be known. Once these are known, the approximating function defines the field variable throughout the assemblage of elements.

B. Advantages of FEM

The FEM is based on the concept of discretization. Nevertheless as either a variational or residual approach, the technique recognizes the multi dimensional continuity of the body not only does the idealizations portray the body as continuous but it also requires no separate interpolation process to extend the approximate solution to every point within the continuum. Despite the fact that the solution is obtained at a finite number of discrete node points, the formation of field variable models inherently provides a solution at all other locations in the body. In contrast to other variational and residual approaches, the FEM does not require trail solutions, which must all, apply to the entire multi dimensional continuum. The use of separate sub-regions or the finite elements for the separate trial solutions thus permits a greater flexibility in considering continuum of the shape.

Some of the most important advantages of the FEM derive from the techniques of introducing boundary conditions. This is another area in which the method differs from other variational or residual approaches. Rather than requiring every trial solution to satisfy the boundary conditions, one prescribes the conditions after obtaining the algebraic equations for assemblage.

VII. General Procedures to Create an Involute Curve

The sequence of procedures employed to generate the involute curve are illustrated as follows: -

1. Set up the geometric parameters Number of teeth Diametric Pitch, Pressure angle, Pitch diameter, Face width, and Helix angle.
2. Create the basic geometry such as addendum, dedendum and pitch circles in support of the gear tooth.
3. Define the involute tooth profile with datum curve by equation using cylindrical coordinate system.
4. Create the tooth solid feature with a cut and extrusion. Additional helical datum curves are also required in this step to sweep helical

gear teeth.

5. Pattern the tooth around the centre line axis.

The key specifications of geometrical parameters and the helical gear model developed by using the above procedures in Pro/Engineer are shown in and Table1

TABLE -2 EXPERIMENTAL RESULTS

Comparison Of Von – Misses Stresses of Modified Gear and Normal Gear	
Gear Type	Von Misses Stresses
Normal Helical Gear	2.0051 MPa
Modified Helical Gear	12.49 MPa

VIII. CONCLUSION

This study is focused on geometry of the teeth to calculate their effect of contact stresses in helical gears. Gear contact analysis is done by using number of formulae and evaluating equations to determine the maximum stress values in analytical method. Parametric study of the helical gear is done by varying the geometrical profile of the gear tooth by using CATIAV5 design software. Three dimensional model is done for both designs and contact stresses of those gears are taken and compared their resultant stresses occurred and deformation at the contact area is taken into account to determine the optimum design. According to the results obtained from the ansys It is concluded that the stresses acting on contact area of a modified helical gear is high when compared with the standard helical gear. This concludes that the project gives you a failure theory such that no change should be done to the standard gear tooth profile.

IX. REFERENCES

1. Yonatan, F., Variable Mesh Stiffness of Spur Gear Teeth Using FEM, M.sc. thesis Department of mechanical Engineering, Addis Ababa University.
2. .Vijayarangan, S., and Ganesan, N., A Static Analysis of Composite Helical Gears Using Three-dimensional Finite Element Method, Computers & Structures, 49,pp.253-268,1993.
3. .Rao, C.M., and Muthuveerappan G., Finite Element Modeling and Stress Analysis of Helical Gear, Teeth, Computers & structures, 49,pp.1095-1106, 1993.
4. Singiresu S. Rao "The Finite Element Method in Engineering".