A Review on Analysis of spur gear in two stage planetary gearbox using finite element approach

1Rupali R. Kaware, 2R. A. Lekurwale, 3S. P. Untawale, 4K. R. Kaware

1,2,3 M.Tech (CAD/CAM), DMIET&R, Wardha
4 GHRCEM Amravati
Email: 1rupali.kaware@hotmail.com, 2rahul.lekurwale2011@gmail.com, 4kiran.kaware@gmail.com

Abstract: In this paper we have seen the finite element method for analysis of spur gear in two stage planetary gearbox. Our project is aimed to carry out design and analysis of two stage spur planetary gearbox2P17. Design is done by doing design calculations analytically according to standard design procedure. Using these analytical values, modelling of gears on Pro-E / CATIA V5R20 modelling software is done. Later on static structural analysis is done by using ANSYS software with finite element approach.

Keywords: Spur gear, static structural analysis

I. INTRODUCTION

A gearbox is a collection of mechanical components that deliver maximum power from an engine by managing a series of gear ratios that in turn operate a transmission. In this paper, analysis of the characteristics of spur gears in a gearbox will be studied using Finite Element Method. Gear analysis was performed using analytical methods, which required a number of assumptions and simplifications. In this paper, bending stress analysis will be performed, while trying to design spur gears to resist bending failure of the teeth, as it affects 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 is studied. This has improved gear analysis and computer simulations. Numerical methods can potentially provide more accurate solutions since they normally require much less restrictive assumptions. The finite element method is very often used to analyze the stress state of an elastic body with complicated geometry, such as a gear.

II. LITERATURE REVIEW:

The review mainly focuses on Finite element approach in analysis of spur gear.

[1] Yi Guo and Robert G. Parker worked on study of the nonlinear tooth wedging behavior and its correlation with planet bearing forces by analyzing the dynamic response of an example planetary gear. The results show significant impact of tooth wedging on planet bearing forces for a wide range of operating speeds. To develop a physical understanding of the tooth wedging mechanism, connections between planet bearing forces and tooth forces are studied by investigating physical forces and displacements acting throughout the planetary gear. A method to predict tooth wedging based on geometric interactions is developed and verified. The major causes of tooth wedging relate directly to translational vibrations caused by gravity forces and the presence of clearance-type nonlinearities in the form of backlash and bearing clearance.

[2] Zhipeng Feng et al. have studied Fault diagnosis of planetary gearboxes. They proposed a simple yet effective method to diagnose planetary gearbox faults based on amplitude and frequency demodulations. They use the energy separation algorithm to estimate the amplitude envelope and instantaneous frequency of modulated signals for further demodulation analysis, by exploiting the adaptability of Teager energy operator to instantaneous changes in signals and the fine time resolution. With the proposed method, both the wear and chipping faults can be detected and located for a sun gear of the planetary gearbox test rig.

[3] A. Kahraman et al. had a main objective of study is to investigate the dynamic effects on gear stresses as a function of gear rim thickness parameters and the number of planets in the system. A deformable body dynamic model is used to simulate a typical automotive automatic transmission planetary unit. A new rim thickness parameter will be introduced that takes into account the size of the gears. The model will be used to quantify the impact of the gear rim flexibilities on dynamic gear stresses. The relationship between the bending modes of the gears and the number of planets in the system is also demonstrated quantitatively.

[4] Chien-Hsing Li et al. worked on batch module called integration of finite element analysis and optimum design by taking gear systems as testing examples. A simple and practical method was
developed, by which this module was enabled to search for contact nodes and elements and to automatically define the contact surfaces for contact analysis. The module will automatically construct the geometrical model, analyze contact stress and solve for the optimal solutions when gearing parameters are input. The results are expected to enhance the technology of gear system design.

[5] J.R. Gom à Ayats had presented a methodology for the kinematic analysis of complex gear trains comprised of planetary gear trains. It is based on the utilisation of hypergraphs, which provides a novel view into the analysis of this type of power trains. The method presented here provides a way to define the kinematic relations existing between all the branches of a mechanism, including branches with a variable speed ratio.

[6] Je`ro`meBruye et al. had evaluated the impact of tolerance on gear quality, designers need to simulate the influences of tolerance with respect to the functional requirements. To do so, they use AGMA or ISO tables, or they perform experimentations. They have proposed an approach to analyze the tolerances.

[7] Zhipeng Fenga and Ming J. Zuohad worked on the spectral structure of planetary gear system vibration signals is helpful to fault diagnosis of planetary gearboxes. Explicit equations for calculating the characteristic frequency of local and distributed gear fault are deduced. The theoretical derivations are validated using both experimental and industrial signals. According to the theoretical basis derived, manually created local gear damage of different levels and naturally developed gear damage in a planetary gearbox can be detected and located.

[8] Aida Rezaie and Azzedine Dadouche studied the main prognostics and health management (PHM) approaches, describes a newly designed gearbox experimental facility and analyses preliminary data for gear prognosis. The rig is based on the accessory gearbox of the GE J85 turbojet engine, which has been slightly modified and reconfigured to replicate real operating conditions such as speeds and loads. Defect to failure tests (DTFT) have been run to evaluate the performance of the rig as well as to assess prognostic metrics extracted from sensors installed on the gearbox casing (vibration and acoustic). The paper also details the main components of the rig and describes the various challenges encountered. Successful DTFT results were obtained during an idle engine performance test and prognostic metrics associated with the sensor suite were evaluated and discussed.

[9] Daniele Vecchiato had considered an isostatic planetary gear train. This mechanism is capable of self-regulation for compensation of various errors of alignment. Localization of bearing contact and application of tooth contact analysis (TCA) are provided. Trajectory of the floating member and function of transmission errors are obtained for several errors of alignment.

III. SPECIFICATION OF THE PROBLEM

In this project work it is proposed to find the failure in existing gearbox

Existing Design of Gears in Two Stage Planetary gearbox named as 2P17 having following problems due to which failure occurs

1. Root safety not achieved
2. Flank safety not achieved
3. Undercut occurred in Gear 1
4. Face width value is not sufficient.

Every gear tooth acts as a cantilever. If the total repetitive dynamic load acting on the gear tooth is greater, then the gear tooth will fail in bending ie gear tooth will break.

In order to avoid such failure, the module and face width of the gear is adjusted so that the beam strength is greater than the dynamic load.

IV. MODELING OF SPUR GEAR

For performing stress analysis of gears, one has to first make 3D model of it. It is a well known fact that preparing the model of the part like a gear is very difficult and time consuming in analysis software like ANSYS. However, one can make the model in any other third party CAD software and bring the solid model to ANSYS for doing the analysis. And the software used in this project for 3D modelling of spur gear is Pro-E.

V. FINITE ELEMENT ANALYSIS OF SPUR GEAR:

Finite element modeling is described as the representation of the geometric model in terms of a finite number of elements and nodes. It is actually a numerical method employed for the solution of structures or a complex region defining a continuum. Solutions obtained by this method are rarely exact.
However, errors in the approximate solution can be minimized by increasing the number of equations till the desired accuracy obtained. This is an alternative to analytical methods that are used for getting exact solution of analysis problems. The solution of general problem by finite element method always follows an orderly step-by-step process.

In the element method the problem is formulated in two stages:

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

2. The system formulation- It is the formulation of the stiffness and loads of the entire structure.

VI. BASIC STEPS IN THE FINITE ELEMENT METHOD

1. 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.

2. 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.

3. 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.

4. 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.

5. 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.

6. 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.

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

8. 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.

9. 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 and Choleky’s factorization are most commonly used methods.

10. Calculation of stresses or stress resultants

VII. CONCLUSION

The parametric model is capable of creating spur gears with different modules and number of teeth by modifying the parameters and regenerating the model. This study provides a sound foundation for future studies on contact stresses. The model is applied onto commercial FEA software ANSYS. Simulation results were compared and confirmed by the theoretical calculation data. According to these results, we can draw the conclusion; it was found out that the numerically obtained values of stress distributions were in good agreement with the theoretical results.

REFERENCES


