International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 2, No. 3, July 2013

Research Paper

INVESTIGATION OF CONTACT STRESS IN SPUR GEAR USING LEWIS EQUATION AND FINITE ELEMENT METHOD

Ashish V Kadu[1]* and Sanjay S Deshmukh[1]

Corresponding Author: **Ashish V Kadu, ✉ashishkadu22@gmail.com*

This work deals with the characteristics of involutes gear system including contact stresses, bending stresses and the transmission error of the gear in mesh the gear system of the unity industry for this work. To estimate the transmission error in the actual gear system which arises because of a irregular tool geometry or imperfect geometry or imperfect mounting the characteristic of the involutes spur gear are analyzed by using finite element method. The contact stresses are examined by using 2D FEM Model. And the bending stresses in the tooth root are examined by using 3D FEM Model. The conventional method of calculating gear contact stress using Hertz's theory for verification by 2D FEM analyzer using ANSYS, in later investigation the stiffness relationship between two contact area is usually established using a spring place between source and target surfaces for the contact generation between two gears. The stresses are compared with theoretical result. This work also considered static transmission error and analysis of load shearing method using displacement vector and the effect of this error in the actual transmission power of the mesh gear.

Keywords: Component, Formatting, Style, Styling, Insert (key words)

INTRODUCTION

Gears are essential to the global economy and are used in nearly all application where the power transmission is required such as automobiles, industrial equipment airplanes helicopters and marine vessels. Frequency of product model changeover, also called time-based competition has become a character feature of global manufacturing of new product development in automotive aerospace and other industries. This forces gear manufacturer to respond with improved gear. Simultaneously, current trends in engineering globalization required research to revisit various normalized standard to determine their common fundamentals and

[1] Department of Mechanical Engineering, Prof. Ram Meghe Institute of Technology & Research, Badnera, Amravati, India.

those approaches needed to identify “best practices” in industries.

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. Noise reduction in gear pairs is especially critical in the rapidly growing field of office-automation equipment as the office environment is adversely affected by noise, and machines are playing an ever-widening role in that environment. Ultimately, the only effective way to achieve gear noise reduction is to reduce the vibration associated with them& hence the transmission inaccuracy of gear. The reduction of noise through vibration control can only be achieved through research efforts by specialists in the field. However, a shortage of these specialists exists in the newer, lightweight industries in Japan mainly because fewer young people are specializing in gear technology today and traditionally the specialists employed in heavy industries tend to stay where they are.

The prime source of vibration and noise in a gear system is the transmission error between meshing gears. Transmission error is a term used to describe or is defined as the differences between the theoretical and actual positions between a pinion (driving gear) and a driven gear. It has been recognized as a main source for mesh frequency excited noise and vibration. With prior knowledge of the operating conditions of the gear set it is possible to design the gears such that the vibration and noise is minimized.

Transmission error is usually due to two main factors. The first is caused by manufacturing inaccuracy and mounting errors. Gear designers often attempt to compensate for transmission error by modifying the gear teeth. The second type of error is caused by elastic deflections under load. Among the types of gearbox noise, one of the most difficult to control is gear noise generated at the tooth mesh frequency.

Transmission error is considered to be one of the main contributors to noise and vibration in a gear set. This suggests that the gear noise is closely related to transmission error. If a pinion and gear have ideal involute profiles running with no loading torque they should theoretically run with zero transmission error. However, when these same gears transmit torque, the combined torsional mesh stiffness of each gear changes throughout the mesh cycle as the teeth deflect, causing variations in angular rotation of the gear body. Even though the transmission error is relatively small, these slight variations can cause noise at a frequency which matches a resonance of the shafts or the gear housing, causing the noise to be enhanced. This phenomenon has been actively studied in order to minimize the amount of transmission error in gears.

In this thesis, first, the finite element models and solution methods needed for the accurate calculation of two dimensional spur gear contact stresses and gear bending stresses were determined by using ANSYS. Then, the contact and bending stresses calculated using

ANSYS were compared to the results obtained from existing methods. The purpose of this thesis is to develop a model to study and predict the transmission error model including the contact stresses, and the torsional mesh stiffness of gears in mesh using the ANSYS software package based on numerical method. The aim is to reduce the amount of transmission error in the gears, and thereby reduce the amount of noise generated.

Objectives of the Research

In spite of the number of investigations devoted to gear research and analysis there still remains to be developed, a general numerical approach capable of predicting the effects of variations in gear geometry, contact and bending stresses, tensional mesh stiffness and transmission errors. The objectives of this thesis are to use a numerical approach to develop theoretical models of the behavior of spur gears in mesh, to help to predict the effect of gear tooth stresses and transmission error. The main focus of the current research as developed here is:

- To develop and to determine appropriate models of contact elements, to calculate contact stresses using ANSYS and compare the results with HERTZIAN theory.
- To generate the profile of spur gear teeth and calculate of gear bending stress using Lewis equation and hence check the feasibility of modified gear profile.
- To compare the static transmission errors of slandered and modified profile of the gear teeth.

Focus of the Work

- Stress analysis such as prediction of contact stress and bending stress.
- Prediction of transmission efficiency.
- Finding the natural frequencies of the system before making the gears.
- Performing vibration analyses of gear systems.
- Evaluating condition monitoring, fault detection, diagnosis, and prognosis, reliability and fatigue life.

LITERATURE REVIEW AND BACKGROUND

There has been a great deal of research on gear analysis, and a large body of literature on gear modeling has been published. The gear stress analysis, the transmission errors, the prediction of gear dynamic loads, gear noise, and the optimal design for gear sets are always major concerns in gear design. Errichello (1979) and Ozguven and Houser (1988) survey a great deal of literature on the development of a variety of simulation models for both static and dynamic analysis of different types of gears. The first study of transmission error was done by Harris (1958). He showed that the behavior of spur gears at low speeds can be summarized in a set of static transmission error curves. In later years, Mark (1978 and 1979) analyzed the vibratory excitation of gear systems theoretically. He derived an expression for static transmission error and used it to predict the various components of the static transmission error spectrum from a set of measurements made on a mating pair of spur gears. Kohler and Regan (1985) discussed the derivation of gear transmission error from pitch error transformed to the frequency domain. Kubo *et al.* (1991) estimated the transmission error of cylindrical involutes gears using a tooth contact pattern. The current literature reviews also attempt to classify gear model into groupings with particular relevance to the research Abbreviations and Acronyms.

Spur Gear Failures

Bending Fatigue: This common type of failure which is a slow, progressive failure caused by repeated loading.

Figure 1: Bending Fatigue

Contact Fatigue: In another failure mode, called contact or Hertzian fatigue, repeated stresses cause surface cracks and detachment of metal fragments from the tooth contact surface.

Wear in Gear: Gear tooth surface wear involves removal or displacement of material due to mechanical, chemical or electrical action.

Scuffing in Gear: Scuffing is a transfer of metal from the surface of one tooth to that of another tooth.

Figure 2: Contact Fatigue

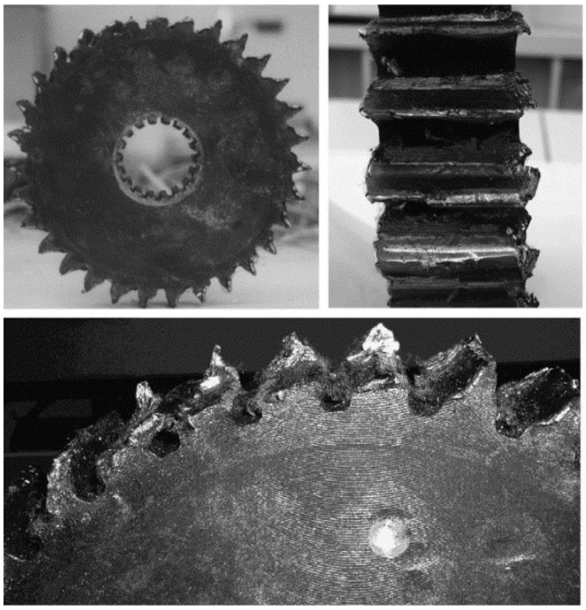
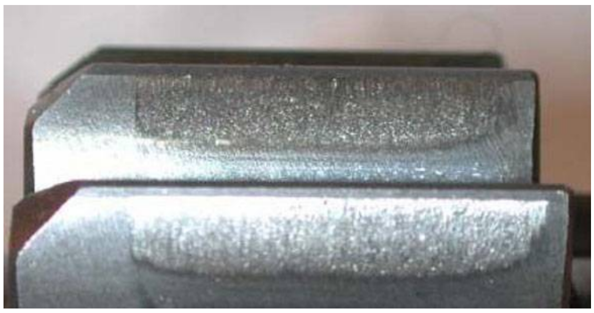


Figure 3: Wear in Gear

Figure 4: Scuffing in Gear



FEM ANALYSIS

Introduction

The finite element method is a numerical technique, well suited to digital computers, which can be applied to solve problems in solid mechanics, fluid mechanics, heat transfer and vibrations. The procedures to solve problems in each of these fields is similar; however this discussion will address the application of finite element methods to solid mechanics problems. In all finite element models the domain (the solid in solid mechanics problems) is divided into a finite number of elements. These elements are connected at points called nodes. In solids models, displacements in each element are directly related to the nodal displacements. The nodal displacements are then related to the strains and the stresses in the elements. The finite element method tries to choose the nodal displacements so that the stresses are in equilibrium (approximately) with the applied loads. The nodal displacements must also be consistent with any constraints on the motion of structure.

The finite element method converts the conditions of equilibrium into a set of linear algebraic equations for the nodal displacements. Once the equations are solved, one can find the actual strains and stresses in all the elements. By breaking the structure into a larger number of smaller elements, the stresses become closer to achieving equilibrium with the applied loads. Therefore an important concept in the use of finite element methods is that, in general, a finite element model approaches the true solution to the problem only as the element density is increased.

There are a number of steps in the solution procedure using finite element methods. All finite element packages require the user to go through these steps in one form or another.

Specifying Geometry: First the geometry of the structure to be analyzed is defined. This can be done either by entering the geometric information in the finite element package through the keyboard or mouse, or by importing the model from a solid modeler like Pro/ ENGINEER.

Specify Element Type and Material Properties: Next, the material properties are defined. In an elastic analysis of an isotropic solid these consist of the Young's modulus and the Poisson's ratio of the material.

Mesh the Object: Then, the structure is broken (or meshed) into small elements. This Involves defining the types of elements into which the structure will be broken, as well as specifying how the structure will be subdivided into elements (how it will be meshed). This subdivision into elements can either be input by the user or, with some finite element programs (or add-ons) can be chosen automatically by the computer based on the geometry of the structure (this is called auto meshing).

Apply Boundary Conditions and External Loads: Next, the boundary conditions (e.g., location of supports) and the external loads are specified.

Generate a Solution: Then the solution is generated based on the previously input parameters.

Postprocessing: Based on the initial conditions and applied loads, data is returned after a solution is processed. This data can be viewed in a variety of graphs and displays.

Refine the Mesh: Finite element methods are approximate methods and, in general, the accuracy of the approximation increases with the number of elements used. The number of elements needed for an accurate model depends on the problem and the specific results to be extracted from it. Thus, in order to judge the accuracy of results from a single finite element run, you need to increase the number of elements in the object and see if or how the results change.

Interpreting Results: This step is perhaps the most critical step in the entire analysis because it requires that the modeler use his or her fundamental knowledge of mechanics to interpret and understand the output of the model. This is critical for applying correct results to solve real engineering problems and in identifying when modeling mistakes have been made (which can easily occur).

The eight steps mentioned above have to be carried out before any meaningful information can be obtained regardless of the size and complexity of the problem to be solved. However, the specific commands and procedures that must be used for each of the steps will vary from one finite element package to another. The solution procedure for ANSYS is described in this tutor. Note that ANSYS (like any other FEM package) has numerous capabilities out of which only a few would be used in simple beam problems.

Limitations of Finite Element Methods

- Finite element methods are extremely versatile and powerful and can enable designers to obtain information about the behavior of complicated structures with almost arbitrary loading.
- In spite of the significant advances that have been made in developing finite element packages, the results obtained must be carefully examined before they can be used. This point cannot be overemphasized.
- The most significant limitation of finite element methods is that the accuracy of the obtained solution is usually a function of the mesh resolution. Any regions of highly concentrated stress, such as around loading points and supports, must be carefully analyzed with the use of a sufficiently refined mesh. In addition, there are some problems which are inherently singular (the stresses are theoretically infinite). Special efforts must be made to analyze such problems.
- An additional concern for any user is that because current packages can solve so many sophisticated problems, there is a strong temptation to "solve" problems without doing the hard work of thinking through them and understanding the underlying mechanics and physical applications. Modern finite element packages are powerful tools that have become increasingly indispensible to mechanical design and analysis. However, they also make it easy for users to make big mistakes.
- Obtaining solutions with finite element methods often requires substantial amounts of computer and user time. Nevertheless, finite element packages have become increasingly indispensable to mechanical design and analysis.

Tools in Finite Element Analysis

Pre-Processing: The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements, "connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical preprocessor to assist in this rather tedious chore. Some of these preprocessors can overlay a mesh on a pre-existing CAD model, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

Solver: The dataset prepared by the pre-processor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations

$$K_{ij}u_i = f_i$$

where, K_{ij} = *Structural Stiffness*

u_i and f_i = The displacements and externally applied forces at the nodal points.

The formation of the *K* matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. The static analysis of gear can be carried out exactly, and the equations of even complicated gear can be assembled in a matrix form amenable to numerical solution. This approach, sometimes called matrix analysis, provided the foundation of early FEA development. Matrix analysis of trusses operates by considering the stiffness of each truss element one at a time, and then using theseto determine the forces that are set up in the truss elements by the displacements of the joints, usually called nodes infinite element analysis. Then noting that the sum of the forces contributed by each element to a node must equal the force that is externally applied to that node, we can assemble a sequence of linear algebraic equations in which the nodal displacements are the unknowns and the applied nodal forces are known quantities. These equations are conveniently written in matrix form, which gives the method its name:

$$\begin{bmatrix} K_{11} & K_{12} & \dots & K_{1n} \\ K_{21} & K_{22} & \dots & K_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ K_{n1} & K_{n2} & \dots & K_{nn} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ u_n \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \\ \vdots \\ f_n \end{Bmatrix}$$

The K_{ij} coefficient array is called the global stiffness matrix, with the *ij* component being physically the influence of the jth displacement on the ith force. The matrix equations can be abbreviated as:

$$K_{ij}u_j = f_i \quad \text{or} \quad Ku = f$$

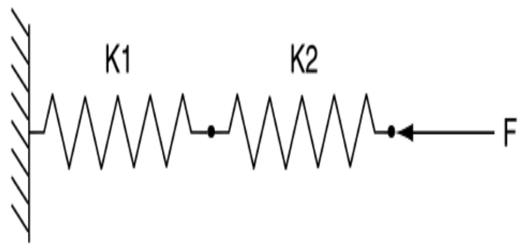
using either subscripts or boldface to indicate vector and matrix quantities. Either the force externally applied or the displacement is known at the outset for each node, and it is impossible to specify simultaneously both an arbitrary displacement and a force on a given node. These prescribed nodal forces and displacements are the boundary conditions of the problem. It is the task of analysis to determine the forces that accompany the imposed displacements, and the displacements at the nodes where known external forces are applied.

Solver Step Tooth Stiffness: An important parameter that must be known in order to determine tooth engagement is the stiffness of a pair of mating internal and external spline teeth. Equation shows the general relationship for a linear spring, where the force is a function of deflection *u*, and the spring rate, or stiffness *K*.

$F = KU$

Because a single spline tooth is an elastic body, it may be treated as a linear spring. The force applied, then determines the deflection uniquely. The stiffness of a single pair of mating spline teeth can be represented as the combination oftwo springs in series. As shown in Figure 5, each spring represents one tooth.

Figure 5: Series Combination of Two Spring



Both springs transmit the applied force to the frame, since they are in series. The total deflection dtotal, is the sum of the two spring deflections, u_1 and u_2. Using Equation the equivalent spring constant of the two teeth in series may be derived from:

$F = K_1 u_1 = K_2 u_2$

Substituting the equivalent stiffness for *K* and the total deflection for μ gives

$F = Keq\ u_1 = Keq\ u_2$

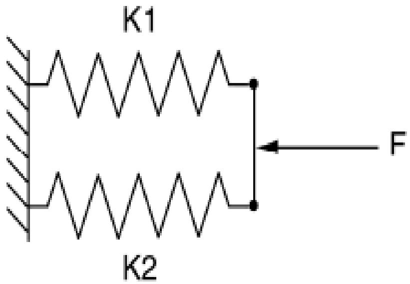
Solving for Keq_s gives

$$Keq_s = \frac{F}{\delta_1 + \delta_2} = \frac{F}{\frac{F}{K_1} + \frac{F}{K_2}} = \frac{1}{\frac{1}{K_1} + \frac{1}{K_2}} = \frac{K_1 K_2}{K_1 + K_2}$$

Because the sum of K_1 and K_2 is in the denominator, Keq_s will always be less than the sum of K_1 and K_2.

For Engagement of Tooth: As sequential teeth engage in a spline coupling, the equivalent stiffness can be determined by the number of teeth that are in contact. Figure 6 represents a pair of linear springs in a parallel configuration.

Figure 6: Parallel Combination of Two Springs



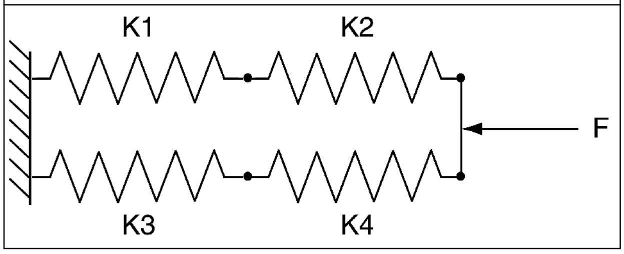
The equivalent stiffness of two springs in parallel is the sum of K_1 and K_2, which is shown in Equation. This is a simplified model of two pairs of spline teeth in contact, sharing the load *F*:

$Keq = K_1 + K_2$

K_1 represents the Keqs of Tooth Pair No. 1 and K_2 is the Keq_s of Tooth Pair No. 2.

By combining Figures 5 and 6, the engagement of two pairs of spline teeth would look like Figure 7. The stiffness of each mating

Figure 7: Series and Parallel Combination of the Spring



internal and external pair adds in series, resulting in an equivalent stiffness Keq_s. The equivalent tooth stiffness for both tooth pairs are then added in parallel.

K_1 and K_3 represent the stiffness of the internal teeth, while K_2 and K_4 represent the stiffness of the mating external teeth. The corresponding equivalent stiffness of the seri,es-parallel combination, Keq_{sp}, is calculated by the following equation

$$Keq_{sp} = \frac{K_1 K_2}{K_1 + K_2} + \frac{K_3 K_4}{K_3 + K_4}$$

Postprocessing: In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. A typical postprocessor display overlay colored contours representing stress levels on the model, showing a full field picture similar to that of photo elastic or moire experimental results.

Contact Analysis

Contact Stress: When two bodies having curved surfaces are pressed together, point or line contact changes to area contact and the stress developed in the two bodies are three dimensional. Contact-stress problem arise in the contact of a wheel and a rail, in automobile valve cams tappets, in the meting teeth, and in the action of rolling bearings. Typical failures are seen as crack, pits or flanking in the surface material.

In general, there are three basic types of contact modeling application as far as practicalapplication is concerned.

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.

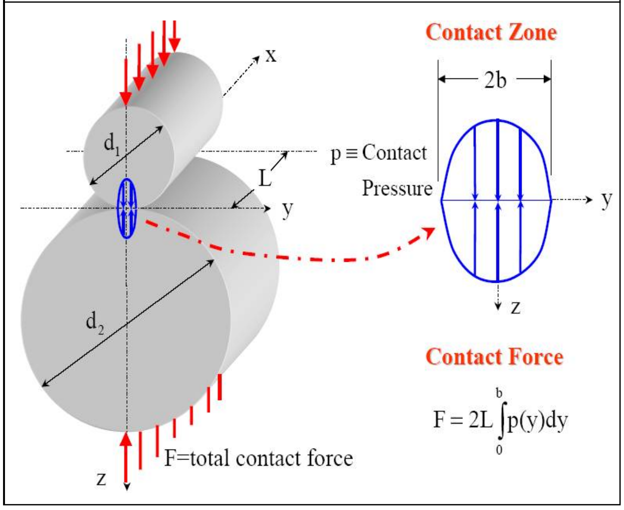
Point-to-Surface Contact: The exact location of the contacting area may not beknown beforehand. These types of contact problems allow large amounts of deformation and relative sliding. Also, opposing meshes do not have to have the same discrimination or a compatible mesh. Point to surface contact was used in this chapter.

Surface-to-surface contact is typically used to model surface-to-surface contact applications of the rigid-to-flexible classification. There are some difficult while dealing with contact problems many difficulties. First, the actual region of contact between deformable bodies in contact is not known until the solution has been obtained. Depending on the loads, materials, and boundary conditions, along with other factors, surfaces can come into and go out of contact with each other in a largely unpredictable manner. Secondly, most contact problems need to account for friction. The modeling of friction is very difficult as the friction depends on the surface smoothness, the physical and chemical properties of the material, the properties of any lubricant that might be present in the motion, and the temperature of the contacting surfaces. There are several friction laws and models to choose from, and all are nonlinear. Frictional response can be chaotic, making solution convergence difficult (ANSYS).

The most general case of contact stress occurs when each containing body has a double radius of curvature, i.e., when the radius in the plane of the rolling is different from the radius in a particular plane, both plane taken planes through the axis of the contacting force. The stress determine from these are also known as Hertzian stresses.

Contact Analysis of Two Cylinders: Consider two solid cylinder of diameter d_1 and d_2 of length *l*, press together with force *F*. As shown in Figure 8 the area of contact is a narrow rectangular of width 2*b* and length *l*, and the pressure distribution is elliptical.

Figure 8: Contacts of Two Cylinder



The half width is given by the equation:

$$b = \sqrt{\frac{2F}{\pi L} \frac{(1 - v_1^2)/E_1 + (1 - v_2^2)/E_2}{1/d_1 + 1/d_2}}$$

the maximum pressure is given by-

$$p_{\max} = \frac{2F}{\pi b L}$$

These equation can be applied to a cylinder and plane surface such as rail by making *d* = infinity for the plane surface the equation can also be applied to internal cylinder by making *d* negative.

Along *X* axis

$$\sigma_x = -2v \cdot p_{\max}\left(\sqrt{1 + \frac{z^2}{b^2}} - \frac{z}{b}\right)$$

Along *Y* axis

$$\sigma_y = -p_{\max}\left[\left(2 - \frac{1}{1 + \frac{z^2}{b^2}}\right)\sqrt{1 + \frac{z^2}{b^2}} - 2\frac{z}{b}\right]$$

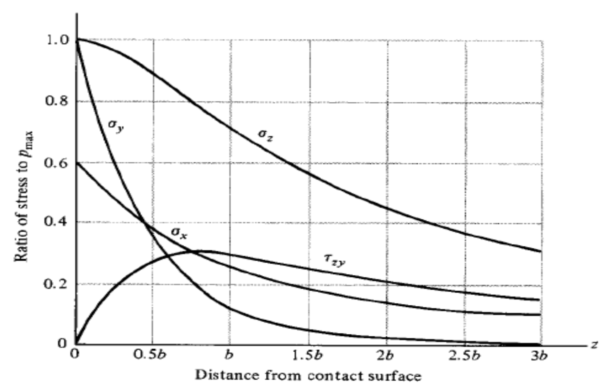
Along *Z* axis

$$\sigma_z = \frac{-p_{\max}}{\sqrt{1 + \frac{z^2}{b^2}}}$$

these three equation are plotted as shown below up to distance of 3*b* below the surface though *Tzy* is not the largest of the three shear stress for all value of *z*/*b*, it is a maximum at about *z*/*b* = 0.75 and larger at that point than point either of the other two shear stress for any for value of *z*/*b*.

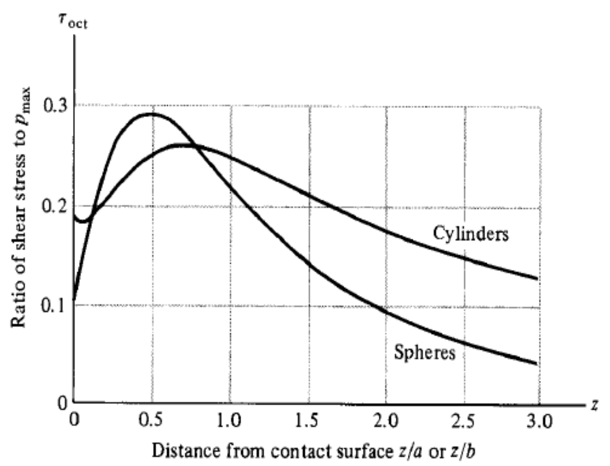
Figure 9 shows the variation of the stress components along the *z*-axis. Note that the maximum shear stress is much less than the maximum contact pressure.

Figure 9: Variation of the Stress Components



Next Figure 10 shows the variation of the octahedral shear stress below the surface for the contacting cylinder. This stress is used some time for failure stress instead of maximum shear stress, to define failure. The octahedral shear stress and maximum shear stress reach their highest value on the curve at the same depth *z*, but the location of appoint is very sensitive to the value of Poisson's ratio.

Figure 10: Variation of the Octahedral Shear Stress



Von Mises stress variation along the *z*-axis. Note that the von Mises stress is much less than the maximum contact pressure.

Hertz (1981) provided the preceding mathematical model of the stress field when the contact zone is free from the shear stress. Another important contact stress case is line of contact withthe friction providing shearing stress on the contact zone such stress are very small with camsandroller follower but flat face follower wheel-rail contact and gear teeth, the stress are evaluated above the Hertzian field.

ANSYS Model for Two Cylinders: In order to verify the FEM contact model procedure, contact between two cylinders was modeled. To reduce computer time, only half cylinders were meshed in the model as shown in Figure 9. The fine meshed rectangular shaped elements were generated near contact areas shown as Figure 10. The dimensions of the elements are based on the half width of the contact area. The contact conditions are sensitive to the geometry of the contacting surfaces, which means that the finite element mesh near the contact zone needs to be highly refined. Finer meshing generally leads to a more accurate solution, but requires more time and system resources. It is recommended not to have a fine mesh everywhere in the model to reduce the computational requirements.

Table 1: ANSYS and HERTZ Results

	ANSYS	Hertz.	Difference
(Pmax), psi	62335	64788	2453

CONCLUSION

It was shown that an FEA model could be used to simulate contact between two bodies accurately by verification of contact stresses between two cylinders in contact and comparison with the Hertzian equations.

Effective methods to estimate the tooth contact stress using a 2D contact stress model and to estimate the root bending stresses using 2D and 3D FEA model are proposed. The analysis of gear contact stress and the investigation of 2D and 3D solid bending stress.

REFERANCES

1. Errichello R (1979), "State-of-Art Review: Gear Dynamics", *Trans. ASME J. Mech. Des.*, Vol. 101, No. 3, pp. 368-372.

2. Harris S L (1958), "Dynamic Load on the Teeth of Spur Gears", *Proc. Instn Meth. Engrs.*, Vol. 172, pp. 87-112.
3. Kohler H and Regan R (1985), "The Derivation of Gear Transmission Error from Pitch Error Records", Proc. Instn. Mech. Engrs., Part C, *Journal of Mechanical Engineering Science*, Vol. 199, No. C3, pp. 195-201.
4. Kubo A *et al.* (1991), "Estimation of Transmission Error of Cylindrical Involute Gears by Tooth Contact Pattern", *JSME Int. J.*, Ser. III, Vol. 34, No. 2, pp. 252-259.
5. Mark W D (1978), "Analysis of the Vibratory Excitation of Gear System: Basic Theory", *J. Acoust. Soc. Am.*, Vol. 63, pp. 1409-1430.
6. Mark W D (1979), "Analysis of the Vibratory Excitation of Gear System II: Tooth Error Representations, Approximations, and Application", *J. Scouts Soc. Am.*, Vol. 66, pp. 1758-1787.
7. Ozguven H N and Houser D R (1988), "Mathematical Models Used in Gear Dynamics", *Sound Vibr.*, Vol. 121, pp. 383-411.
8. Zeping Wei (1989), "Stresses and Deformations in Involute Spur Gears by Finite Element Method".