Computational Analysis of Contact Stresses In Involute Spur Gears Using ANSYS

Authors
D.V. Subbarao¹, Y. Dilip Kumar², T. Tirupati³
¹PG Scholar Mechanical Department MRCET Secunderabad
²Associate Professor Mechanical Department MRCET Secunderabad
³Associate Professor Mechanical Department Bharath Engineering College Hyderabad
Email-v.subbu56@gmail.com, dilipkumar1999@rediffmail.co, tirupatitulasi@yahoo.co.in

ABSTRACT
The present paper investigates computational analysis of the contact stresses involved in an involute gear system using ANSYS simulation software. Gearing is one of the most critical components in mechanical power transmission systems. The computational simulation of contact stresses of two cylinders is carried out to validate the methodology with available analytical method, Hertz contact stress equation, which are originally derived for contact between two cylinders. The simulation of contact problems with ANSYS is carried out with the stiffness relationship between the two contact areas is usually established through a contact element, spring type is placed between the two contacting areas. The results of the two dimensional FEM analyses using ANSYS, a FEM solver software are presented. The results simulated are in agree with the theoretical values calculated using Hertz contact stress equation.

Keywords-ANSYS, contact stress, computational, FEM, gear

INTRODUCTION
Gearing is one of the most critical components in a mechanical power transmission system, and in most industrial rotating machinery. It is possible that gears will predominate as the most effective means of transmitting power in future machines due to their high degree of reliability and compactness. 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.

Designing highly loaded spur 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, along with transmission errors. Gears analyses in the past are performed using analytical methods,
which required a number of assumptions and simplifications. In general, gear analyses are multidisciplinary, including calculations related to the tooth stresses and to tribological failures such as like wear or scoring. 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 [12] and Ozguven and Houser[13] 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 [14]. 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 [15] and [16] 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 [17] discussed the derivation of gear transmission error from pitch error transformed to the frequency domain. Kubo et al [18] estimated the transmission error of cylindrical involute gears using a tooth contact pattern.

In this research paper, preliminary investigation of static contact stresses between two cylinder to validate the methodology that can be implemented for gears in contact resulting in complicated mesh are performed using ANSYS.

**COMPUTATIONAL MODELING OF CONTACT ANALYSIS**

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.

The contact problem is addressed using a special contact element. A number of contact elements were available (two and three dimensional, spring and damper combinations). For the problem in hand, the element to be used is a two-dimensional, the three nodes, and point-to-surface contact element. In the input file, the CONTAC48 element from the ANSYS element library as the contact elements between the two contact bodies shown as Fig.1 is chosen. It is applicable to 2-D geometry, plane strain, plane stress, or axisymmetry situations. The area of contact between two or more bodies is generally not known in advance. It may be applied to the contact of solid bodies for static or dynamic analyses, to problems with or without friction, and to flexible-to- flexible or rigid-to-flexible body contact.
First, to investigate the accuracy of the present method, two circular elastic discs under two-dimensional contact are analyzed, and the computational solutions are compared with that of the Hertz theory. The calculation is carried out under a plane strain condition with a Poisson’s ratio of 0.3 using eight-node isoparametric elements.

Consider two circular discs, A and B, with a radius of R1 = 3 in. and R2 = 3 in. as shown in Fig.2. To reduce the number of nodes and elements and to save more computer memory space, half of the discs are partitioned to the finite element mesh, the number of elements and nodes for each disc is 1766 and 1281, respectively.

In this problem, two steel cylinders are pressed against each other. This model is built based on the Hertz contact stress theoretical problem. The radii were calculated from the pitch diameters of the pinion and gear and other parameters shown in Table.1 and Fig.2. The contact stress of this model should represent the contact stress between two gears. In the input file, first, the geometry of two half cylinders, must be described. Then the geometry areas are meshed. In contact areas a fine mesh is built. The boundary conditions are applied in this model. The loads also are applied four times as four steps. In each step there are a lot of sub-steps. In each sub-step the number of equilibrium iterations was set. The steel material properties have an elastic Young’s modulus of 30,000,000 psi and the Poisson’s ratio was 0.30.

<table>
<thead>
<tr>
<th>Number of teeth</th>
<th>25</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal Module (M)</td>
<td>6 mm</td>
</tr>
<tr>
<td>Addendum Modification coefficient</td>
<td>0</td>
</tr>
<tr>
<td>Normal Pressure Angle</td>
<td>20 degrees</td>
</tr>
<tr>
<td>Face Width (mm)</td>
<td>0.015 M</td>
</tr>
<tr>
<td>Addendum (mm)</td>
<td>1.00 M</td>
</tr>
<tr>
<td>Dedendum (mm)</td>
<td>1.25 M</td>
</tr>
</tbody>
</table>
RESULT OF THE CONTACT STRESS ANALYSIS

The objective of the contact stress analyses is to gain an understanding of the modeling and solution difficulties in contact problems and examine the contact stresses in the gears. In order to verify the FEM contact model procedure, contact between two cylinders is modeled. To reduce computer time, only half cylinders were meshed in the model as shown in Fig.3. The dimensions of the elements are based on the half width of the contact area.

The normal contact stress along the contact surface from the ANSYS solution is shown in Fig.4. Fig.4 (a) and (b) show the distributions of the contact stress along the contact area. The comparison of results from FEM and the Hertzian theoretical formula are shown in Fig.5 in which the two distributions lie very close. The red color line represents the value from the theoretical Hertz equation and the blue color points represent the results from ANSYS. They match very well. It is easy to see the blue color points are on the red curve.
Fig. 4 Normal contact stress along the contact surface

Fig. 5 Contact stress from ANSYS agrees with the Hertz stress

Fig. 6 (a) and (b) show orthogonal shear stress. Figures 3.12 (a) and (b) show maximum shear stresses under the contact areas between two cylinders. The largest orthogonal shear stress lies below the surface at the edge of the contact zone. This was shown in Figure 3.11 (b). The subsurface location of the maximum shear stress can also be seen lying below the surface at the center of the contact zone shown in Figure 3.12 (b). If both materials are steel, it occurs at a depth of about 0.63 a where a is half of the contact length shown in Figure 3.4 and its magnitude is about 0.30 Pmax. The shear stress is about 0.11 Pmax at the surface on the z axis. The subsurface location of the maximum shear stress is believed to be a significant factor in surface-fatigue failure. The theory indicates that cracks that begin below the surface eventually grow to the point that the material above the crack breaks out to form a pit.
CONCLUSION

Finite element modeling of the contact between two cylinders was examined in detail. The finite element method with special techniques, such as the incremental technique of applying the external load in the input file, the deformation of the stiffness matrix, and the introduction of the contact element were used. It was found that initial loading using displacements as inputs was helpful in reducing numerical instabilities.

REFERENCES