IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
ISSN: 2278-1684, PP: 26-31
www.iosrjournals.org

Static Analysis of Spur Gear Using Finite Element Analysis

Mrs.Shinde S.P.[1], Mr. Nikam A.A.[2] , Mr. Mulla T.S.[3]
[1]*(Assistant Professor,Dept. of Mechanical Engg, Dr.J.J.Magdum College Of Engg. Jaysingpur, India)*
[2]*(Assistant Professor,Dept. of Mechanical Engg, Dr.J.J. Magdum College Of Engg. Jaysingpur, India)*
[3]*(Assistant Professor,Dept. of Mechanical Engg, Dr.J.J. Magdum College Of Engg. Jaysingpur, India)*

ABSTRACT : *Gear is one of the most critical component in a mechanical power transmission system, and most industrial rotating machinery. A pair of spur gear teeth in action is generally subjected to two types of cyclic stresses: bending stresses inducing bending fatigue and contact stress causing contact fatigue. Both these types of stresses may not attain their maximum values at the same point of contact fatigue. These types of failures can be minimized by careful analysis of the problem during the design stage and creating proper tooth surface profile with proper manufacturing methods. In general, gear analysis is multidisciplinary, including calculations related to the tooth stresses and to tribological failures such as wear or scoring. 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. First, the finite element models and solution methods needed for the accurate calculation of bending stresses will be determined. Then bending stresses calculated using ANSYS, were compared to the results obtained from existing methods.*

Keywords– *Beam strength, Bending stress, contact stress, Gear analysis using ANSYS, Static load.*

1. Introduction

Gears usually used in the transmission system are also called speed reducer, gear head, gear reducer etc., which consists of a set of gears, shafts and bearings that are factory mounted in an enclosed lubricated housing. Speed reducers are available in a broad range of sizes, capacities and speed ratios. In this paper, analysis of the characteristics of spur gears in a gearbox will be studied using linear 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.

There are two theoretical formulas, which deal with these two fatigue failure mechanisms. One is the Hertz equation, which can be used to calculate the contact stresses and other is the Lewis formula, which can be used to calculate the bending stresses.
The finite element method is capable of providing this information, but the time needed to create such a model is large. In order to reduce the modeling time, a 3D model created in solid modeling software can be used. One such model is provided by PRO-E. PRO-E can generate models of three-dimensional gears easily. In PRO-E, the geometry is saved as a file and then it can be transferred from PRO-E to ANSYS in IGES form. In ANSYS, one can click File > Import > IGES > and check.
The main focus of the paper is:-

1) To develop and to determine bending stresses using ANSYS and compare the results with conventional methods.
2) To generate the profile of spur gear teeth and to predict the effect of gear bending using a three dimensional model and compare the results with those of the Lewis equation.
3) To compare the accuracy of results obtained is ANSYS by varying mesh density

2. Finite Element Analysis

In this finite element analysis the continuum is divided into a finite numbers of elements, having finite dimensions and reducing the continuum having infinite degrees of freedom to finite degrees of unknowns. It is assumed that the elements are connected only at the nodal points.

The accuracy of solution increases with the number of elements taken. However, more number of elements will result in increased computer cost. Hence optimum number of divisions should be taken.

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

2.1 The element formulation

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

2.2 The system formulation

It is the formulation of the stiffness and loads of the entire structure.

3. Basic steps in the finite element method

3.1. 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.

3.2. 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.

3.3. 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.

3.4. 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.

3.5. 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.

3.6. 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.

3.7. Formation of the element loading matrix.

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

3.8. 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.

3.9. 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.

3.10. Calculation of stresses or stress resultants

The nodal displacement values are utilized for calculation of stresses. This may be done for all elements of the continuum or may be limited only to some predetermined elements.

4 Finite Element Models

4.1. The two dimensional models

Fatigue or yielding of a gear tooth due to excessive bending stress is two important gear design considerations. In order to predict fatigue and yielding, the maximum stresses on the tensile and compressive sides of the tooth, respectively, are required. When meshing the teeth in ANSYS, if ―SMART SIZE is used the number of elements near the roots of the teeth are automatically are much greater than in other places. It also indicates that only one tooth is enough for the bending stress analysis for the 3-D model or a 2-D model.

4.2 Three dimensional models

In this section the tooth root stresses and the tooth deflection of one tooth of a spur gear is calculated using an ANSYS model. For the bending stress, the numerical result is compared with the values given by the draft proposal of the standards of AGMA. The element type ―SOLID TETRAHEDRAL 10 NODES 187 was chosen. Because ―SMART SET was chosen on the tool bar there are many more elements near the root of the tooth than in other places. From the Lewis equation if the diameters of the pinion and gear are always kept the same and the number of teeth was changed, the diametric pitch will be changed or the module of gears will be changed. That means that there are different bending strengths between the different teeth numbers.

5 Analysis of Gear

5.1 Theoretical Calculation of Gear

a) Face width =10 m (m=module)

F.S. = 1.5, Y =0.308 for Z_p =18, C.S. = 1.5

$$m = \left\{\frac{60\times10^6}{\pi}\left[\frac{kw\times C_s\times F.S.}{Z_p\times n_1\left(\frac{S_{ut}}{3}\right)\left(\frac{b}{m}\right)\times Y\times C_v}\right]\right\}^{1/3}$$

$$= \left\{\frac{60\times10^6}{\pi}\left[\frac{8\times1.5\times1.5}{18\times1440\times\left(\frac{720}{3}\right)\times(10)\times0.375\times0.308}\right]\right\}^{1/3}$$

$$m = 3.73\approx 4mm$$

b) Beam strength (Sb)

$$S_b = \frac{s_{ut}}{3} = 240\ N/mm2$$

$$S_b = 4\times 40\times 240\times 0.308 = 11827.2\ N$$

c) Static load (M_t)

$$M_t = \frac{60\times10^6\times kw}{2\times\pi\times n_p}$$

$$M_t = \frac{60\times10^6\times8}{2\times\pi\times1440}$$

$$M_t = 53.05\times10^3\ N.mm$$

d) Pitch line velocity:-

$$V = \frac{\pi d_p n_p}{60\times10^3} = 5.42\ m/sec.$$

$$C_v = \frac{3}{3+v} = 0.3559$$

$$P_{eff} = 6210.28\ N$$

$$S_b > P_{eff}$$

It is concluded that the design is safe for static load and dynamic load for bending stresses of beam strength.

5.2 Analysis of Spur Gear Using ANSYS

Step 1:- Import the solid model from pro e:-File-import-IGES

Step 2:-Preference →Structural→ OK.
Step 3:- Preprocessor
Selecting the element type for the gear
Preprocessor → Element type →Add/Edit/Delete→ click on Add in the dialogue box that appears → In Library of Element Types select Solid-Tet 10node 187→Ok
Step 4:- Specifying the material properties
Choose preprocessor→ Material props→ Material Models→ Double click on Structural→ Linear→ Elastic→ Isotropic. The Young's modulus of plain carbon steel is **2.1 E5** and the poisson ratio (PRXY) is **0.3.** Choose preprocessor→ Material props→ Material Models→ Double click on Structural→ Density.
Density of plain carbon steel is **6.8e-6**
Step 5:-Meshing the geometry
Descretize the model into finite elements. Set the element edge length to 30.

a) Meshing→ Size control→ Manual → Global→ Size→ Enter Element Edge Length as 30 → OK
b) Now mesh all the volumes in the geometry

Step 6:-Applying load and constraints
Loads→ Define Loads→ Apply → Structural→ Displacement→ On Areas→ Select the inner surfaces of the key hole→ OK
Step 7:- Solution
Solution→ Solve→ Current LS→ In the prompting that appears on the screen click YES
Step 8:-General Postprocessor
General Post procedure → Plot Results→ in the Plot Deformed Shape dialogue box select Def + un-deformed → General Post procedure→ Plot Results→ Contour Plot→ Nodal Solu→ Stress→ Von mises Stress→ OK

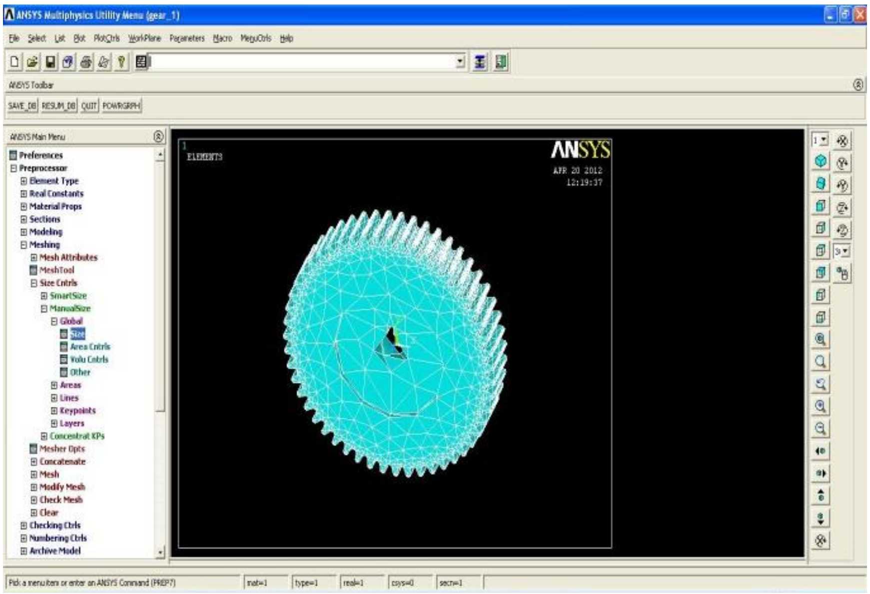

Figure No. 1 Meshing of Gear

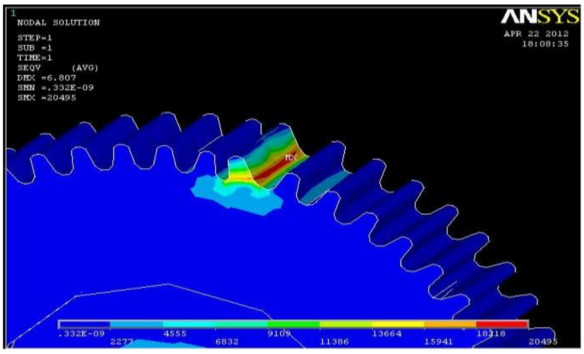

Figure No. 2 Element Edge

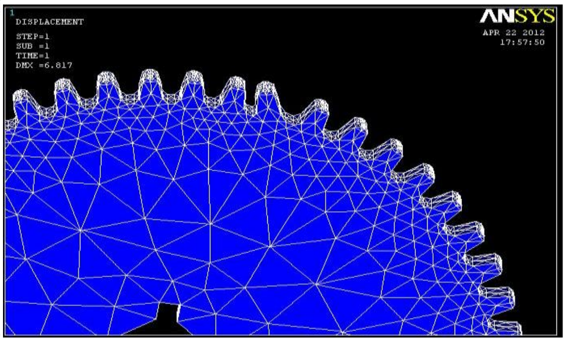

Figure No. 3 General Post procedure

Table 1. Geometry of the gear set

Description	**Symbol**	**Formulae**	**Values**
Number of teeth on pinion	z_p	-	18
Number of teeth on gear	z_g	-	51
Pressure angle	α_n	-	20^0
Module (mm)	m	-	4
Circular pitch (mm)	p	$p = \pi \times m$	12.5663
Pitch circle diameter (mm)	d_p	$d_p = m \times z$	204
Addendum height (mm)	h_a	$h_a = m$	4
Dedendum height (mm)	h_d	$h_d = 1.25m$	5
Addendum circle (mm)	d_a	$d_a = d_p + (2 \times m)$	212
Dedendum circle (mm)	d_d	$d_d = d_p - (2 + \pi/z) \times m$	194
Base circle (mm)	d_b	$d_b = d_p \times \cos \alpha_n$	192
Face width (mm)	F	$F = 10 \times m$	40

Table 2 Comparison of results

Size of Element	Solution by conventional method	ANSYS result	% Accuracy
30	11827.2	11777	99.57
40		11386	96.26
50		13179	89.74

6 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. Sets of gears having the same module and pressure angle can be created and assembled together. It is possible to carry out finite element analysis such as root bending stress and contact stresses between gear teeth pair and effect of root fillet radius on the root stresses.

In this paper, a 3D deformable-body (model) of spur gears is developed. The result is checked with theoretical calculation data. The simulation results have good agreement with the theoretical results, which implies that the deformable-body (model) is correct. 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

[1] V.B.Bhandari ,-Design of Machine Elements, Tata McGraw Hill, New Delhi 2010.
[2] Maitra, G.M, Hand Book of Gear Design, TataMcGraw-Hill, c, 2004.
[3] M.J.Fagan. Finite Element Analysis-Theory and Practices, Longman Group UK Ltd, 1992.
[4] Pankaj Kumar Jena,2008, “Static and dynamic analysis of HCR Spur gear drive using finite element analysis” B.Tech Thesis, Dept. of Mechanical Engineering, NIT Rourkela.
[5] S.S.Rao, The Finite Element Method in Engineering.
[6]Tickoo-Maini,Pro-Engineering Wildfire 3.0,Dreamtech,New Delhi 2007.
[7] Ivana Atanasovska, Vera Nikolic-Stanojlovic, Dejan Dimitrijevic, Dejan Momcilovic, “Finite Element Model for Stress Analysis and Nonlinear Contact Analysis of Helical Gears ”, Institute Kirilo Savic,Serbia, 5/3/2009.
[8] Mike Renfro, “Modelling Gear Teeth in Pro/ENGINEER Wildfire 4.0” Macmillan”, National Kaohsiung University of Applied Sciences, Taiwan, 27/5/2010.
[9] G. Mallesh, Venkatesh, Shankarmurthy H J, Shiva Prasad,“ Parametric analysis of Asymmetric Spur Gear Tooth”, NIT, Durgapur, India, December 17-18, 2009.
[10] Gitin M Maitra, Rourkela, Handbook of Gear Design, 2nd Edn, Tata McGraw-Hill, 1989.