CONTACT STRESS ANALYSIS OF INVOLUTE SPUR GEAR BY FINITE ELEMENT METHOD (FEM)

Shiferaw Damtie and Daniel Tilahun
School of Mechanical & Industrial Engineering, Addis Ababa Institute of Technology, AAU
Corresponding Author's E-mail: daniel.redda@yahoo.com

ABSTRACT

In this paper the contact stress in rolling-sliding contact of involute spur gear and the effect of coefficient of friction was analyzed. To achieve this, first, three dimensional involute spur gear pairs were developed in Solid works 2012 Premium and the 3D model was exported to ANSYS workbench 14.5. Next, the analysis was done using both FEM and Hertizian stress formula. Thus, it is found that the contact stress both in the FEM and analytical method is the similar with minimum variation. In addition, it is found that the contact stress increases with higher coefficient of friction between the areas of contact. Therefore, the analytical method can be applied for gear design of high load carrying capacity effectively.

Keywords: Contact Stress, Involute Spur gear, FEM.

INTRODUCTION

Gears are subjected to rolling-sliding contact, for the spur gear, the contact between two teeth follows the line of action. On the driving wheel it starts at the tooth root and moves towards the top. For the following wheel the movement is reversed, that is from top to root. For the first half of the interaction, when the contact is located between the root and the pitch circle of the driving wheel, the friction force is directed towards the root of the driving tooth, counteracting the rotation. When the contact passes the pitch circle, the friction force change direction and acts with the rolling motion. For the following tooth the friction forces are directed in the opposite direction. Thus for the spur gear, relative motion is constant across the contact width but variable through the interaction.

Gear design is commonly based on the comparison of the maximum tooth-root stress with the permissible bending stress [1]. Their determination depends on a number of different coefficients that allow proper consideration of real working conditions (additional internal and external dynamic forces, contact area of engaging gears, gear's material, surface roughness, etc.). The classical procedures are exclusively based on the experimental testing of the reference gears, and they consider only the final stage of the fatigue process in the gear tooth root, i.e. the occurrence of final failure.

The major drawback with experimental method is that it cannot be undertaken until a prototype exists. In case a design problem occurs it is usually very difficult to rectify. It is also very expensive to perform experimental tests. For these reasons FEM based contact stress analysis has been perceived as an excellent enhancement to the experimental method and applied in many research activities relevant to gear design discussed below.

A lot of research has been carried out on gear modeling and analysis. Gear stress analysis, transmission errors, and prediction of gear dynamic loads, gear noise, and optimal design for gear sets are always major concerns in gear design. Finite element method is widely used for analysis of gear tooth. For example, the effect of rim thickness on gear tooth bending stress by finite element modeling approach was studied [2].

M. Raja Roy et.al., [3] studied the trend of maximum allowable contact pressure on involute pair of spur gear teeth decreases with increase in module and concluded that both analytical and finite element analysis by ANSYS follow the same trend, that is, contact pressure decreases with increase in module.

Ali Raad Hassan [4] analyzed contact stress, considering contact ratio, approach angle, recess angle, contact and length of contact. It was found that stress was more than the correct value of contact stress obtaining from approximating tools. This research was carried out with the help of finite element analysis which could have been difficult to do otherwise. Ten different positions were taken at intervals of three degree and the corresponding ten finite models and finite element analysis was done. On this study it is found that the maximum stress occurs in a single tooth contact position and less stress before and after this particular position. Parveen Kumar, Harsh Raghuvanshi [5] analyse the effect of geometric conditions like gear ratio, normal module and face width on the failure load and beam strength is done. The stress analysis is done by using solid works.

Ravichandra Patchigolla and Yesh P. Singh [2], used Finite Element Method based approach to investigate the effect of the gear rim thickness on the tooth bending stresses in large spur gears. A program was developed using ANSYS Parametric Design Language (APDL), which can generate two dimensional or three dimensional finite element model of 1, 3 or 5 teeth segment with user defined rim thickness value. Models were constrained on the radial sides in the rim portion and also on the nodes located circumferentially along the bottom surface at the rim-web interface. The models are studied for the case of full load acting at Highest Point of Single Tooth Contact (HPSTC). A different meshing approach was developed in the generation of finite element grid.

G. Mallesh, et.al., [6] generated asymmetric spur gear tooth profile for different pressure angles on drive and coast sides and estimate the critical section using C- programming. Bending stress analysis has been performed using finite element analysis with ANSYS software. Comparison of bending stress analysis has been performed for symmetric and asymmetric spur gear tooth at critical section.

Mushin J. Jweeg, et.al. [7] used 2D contact stress FEA model to simulate contact between two bodies accurately by verification of contact stresses between the two gears in contact and concluded that the module has the greater effect on the behaviour of the tooth contact stresses. Decreasing module size leads to increase in the contact stress. Increasing the spur gear design parameters (pressure angle, number of teeth and module) leads to improvement in the tooth strength by increasing the thickness of the critical section which results in increasing the area of tooth critical section and makes it able to withstand higher loads.

ANALYTICAL METHODS AND CONDITIONS

Materials

The list of materials and their characteristics used for the analysis are shown in Table 1 [8].

Table 1: Gear Materials and Characteristics

Material	**Bulk modulus of Elasticity**	**Poisson's ratio**
Chromium-molybdenum alloy steel (SCM 420)	206	0.3
Structural steel	200	0.3

Gear Specifications

The specifications of the pinion and gear used for this study are shown in Table 2. For computational advantage the number of teeth of pinion and gear are made to be equal and the face width is also small.

Table 2: Spur Gear Specification

Parameters	**Pinion**	**Gear**
Material	Structural steel	Structural steel
No. of Teeth	22	22
Module (mm)	3	3
Pitch circle Diam. (mm)	66	66
Face width (mm)	7	7

Modeling

For generation of involute spur gear the parametric equations (Eq.1) were used:

$$\left.\begin{aligned} x(t) &= R\times[\cos(t)+t\times\sin(t)] \\ y(t) &= R\times[\sin(t)-t\times\cos(t)] \end{aligned}\right\} \quad \ldots\ldots\ldots.(1)$$

Where R is pitch radius;

t is angle parameter in radius;
$x(t)$ is a function with respect to parameter t; and
$y(t)$ is a function with respect to parameter t

Using Eq. (1), a two dimensional sketch of the involute spur gear is generated and this sketch is then extruded to a length equal to the face width of the gear which can give a three dimensional involute spur gear.

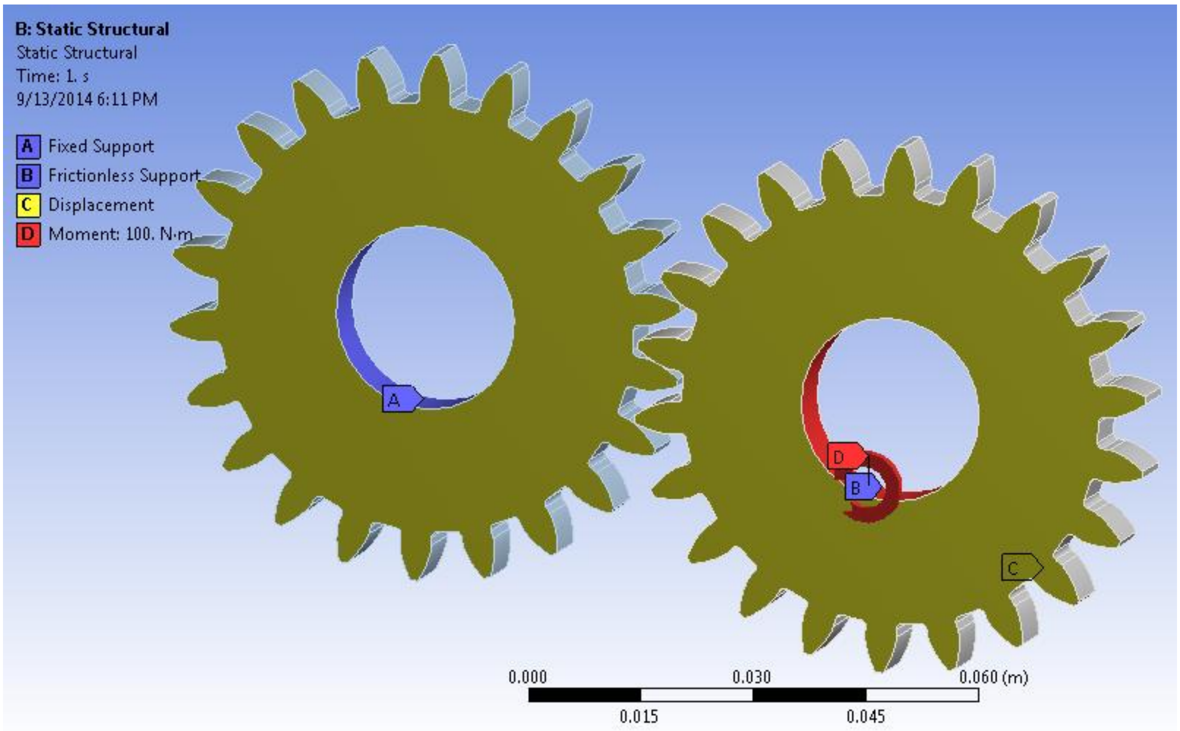

Fig. 1: Assembly of Meshing Pinion and Gear

The same procedure is used for generation of pinion and gear and assembly of meshed pinion and gear with the applied boundary condition is shown in Fig. 1.

Analytical Methods (FEM and Hertzian Formula)

From practical point of view, Finite Element Method is preferable for contact stress analysis of involute spur gear [9, 10, 11].

The contact stress developed in the area of contact of the pinion and gear was calculated analytically by using Finite Element Method-ANSYS Workbench Release 14.5 and Hertz equation. Basically, the number of mesh determines the accuracy of the result and computational time. In order to manage the FEM analysis and accuracy, the research applied fine mesh in the area of interest (contact area) and coarse mesh away from the contact region as shown in Fig. 2.

On the other hand, contact stress analysis using Hertzian equation was performed using a computer programming code- Matlab Release 2010b as shown in Eq. (1).

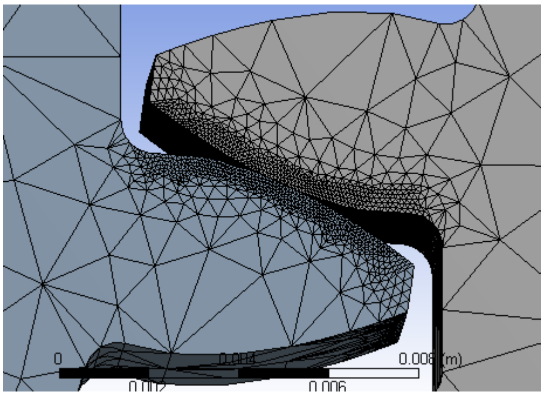

Fig. 2: Fine Mesh Near the Contact of FEM Analysis

In principle, the meshing of two involute teeth have similar with the contact two cylinders or disks. Hence Hertz equation can be used to determine the contact stresses in the mating teeth of gear. Hertzian equation for contact stress in the teeth of mating gears is given by

$$\sigma_c = \sqrt{\frac{F\left(1+\frac{R_1}{R_2}\right)}{R_1 B\pi\left[\frac{(1-\vartheta_1^2)}{E_1}+\frac{(1-\vartheta_2^2)}{E_2}\right]\sin\phi}} \quad ...(2)$$

Where σ_c is the contact stress in mating teeth of spur gear; F is the force; R_1 and R_2 are pitch radii of two mating gears; B is the face width of gears; ϕ is the pressure angle; ϑ_1 and ϑ_2 are the Poisson ratios; and E_1 and E_2, are the moduli of elasticity of two gears in mesh.

The classical theory of contact focused primarily on non-adhesive contact where no tension force is allowed to occur within the contact area, i.e., contacting bodies can be separated without adhesion forces. Several analytical and numerical approaches have been used to solve contact problems that satisfy the no-adhesion condition. Complex forces and moments are transmitted between the bodies where they touch, so problems in contact mechanics can become quite sophisticated. In addition, the contact stresses are usually a nonlinear function of the deformation. To simplify the solution procedure, a frame of reference is usually defined in which the objects (possibly in motion relative to one another) are static.

The bodies interact through surface tractions (or pressures/stresses) at their interface. The following assumptions are made in determining the solutions of Hertzian contact problems: the strains are small and within the elastic limit, the surfaces are continuous and non-conforming (implying that the area of contact is much smaller than the characteristic dimensions of the contacting bodies), each body can be considered an elastic half-space and the surfaces are frictionless.

The contact stress was calculated for different loading conditions which ranges from 10 Nm to 100 Nm for both FEM and analytical method using Hertzian equation.

RESULTS AND DISCUSSION

Results

Fig. 3 show the effect of contact stress on the overall stress (Von Mises Stress) in the gear pair. As it is shown in Fig. 3, the maximum stress is at the contact area where the contact stress is maximum. The contact stress distribution is shown in Fig. 4 which indicates that the contact stress is most critical for failure of gears in such loading condition.

Discussion

The contact stress distribution for various loading condition is given in Table 3 as well as Fig. 4. The error between FEM and Hertzian equation decreases with increasing loading condition and even for smaller loads, the error is tolerable.

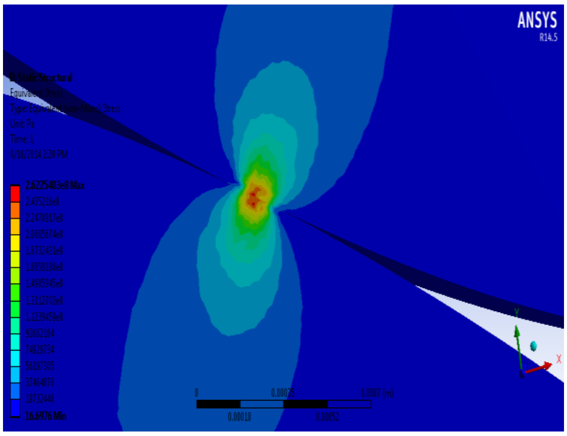

Fig. 3: Equivalent Von Mises Stress Distribution

The influence of coefficient of friction was also studied by taking five different values of coefficients and the contact stress is observed to increase with increasing coefficient of friction. Figure 4 shows the graph of contact stress by Hertizean theory and by FEM for different Material at applied moment of 100Nm.

As shown in Table 4, the contact stress is less when higher strength material is used and the values from the ANSYS workbench and Hertizean formula are agreeable.

Since there is no previous experimental work done with the same parameter the FEM result is compared with Hertizean contact stress. The Hertizean contact stress calculated for the moments 10, 15, 20, 25, 30, 35, 40, 45, 50, 55, 100 Nm are calculated and then for those elements the maximum contact pressures are found from FEM by ANSYS Workbench 4.5.

As seen in Table 3, the error for 100Nm is 1.496 and it would be less for higher moment. The contact stress from Hertizean stress and from ANSYS Workbench is plotted in Fig. 4. The error versus applied moment is plotted as shown in Fig. 9; as it is seen in the figure the error decreases when the load increases.

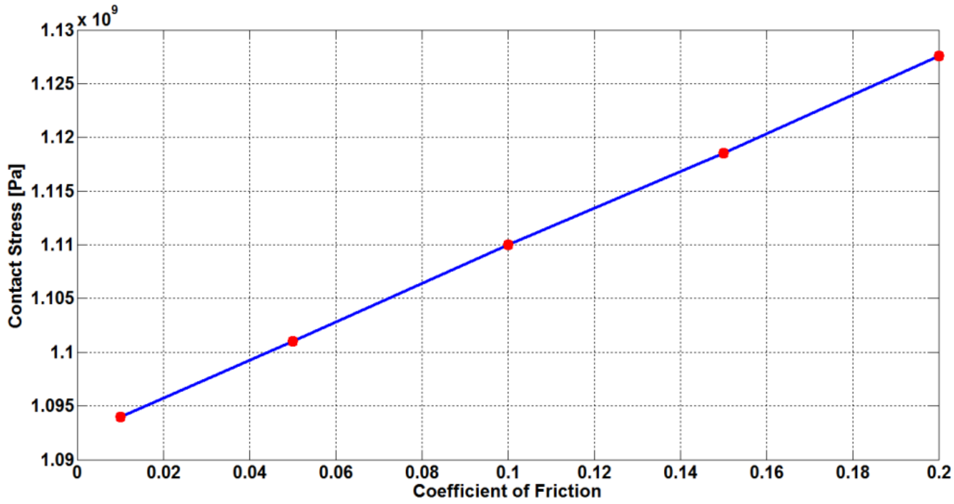

Fig. 4: Influence of Coefficient of Friction on Contact Stress

Table 3:.Comparison of Analytical Contact Stress with FEM

Contact sizing =0.0004m ;Friction coefficient=0.15			
Moment Applied [Nm]	**Contact Stress ANSYS Workbench) [MPa]**	**Hertz Contact Stress [MPa]**	**Error (%)**
10	486.38	518	6.03
15	596.22	634.42	6.02
20	694.31	732.56	5.22
25	778.5	819	4.95
30	853.98	897.2	4.8
35	925.89	969.1	4.46
40	992.78	1036	4.17
45	1056.5	1098.8	3.85
50	1118.5	1158.3	3.48
55	1177.7	1214.8	3.05
100	1662.6	1638.1	1.496

Table 4: Comparison of contact stress by Hertz and FEM

Material	**FEM Contact Stress [MPa]**	**Hertz Contact Stress [MPa]**
Chromium Molybdenum	1679.8	1662.5
Structural steel	1662.6	1638.1

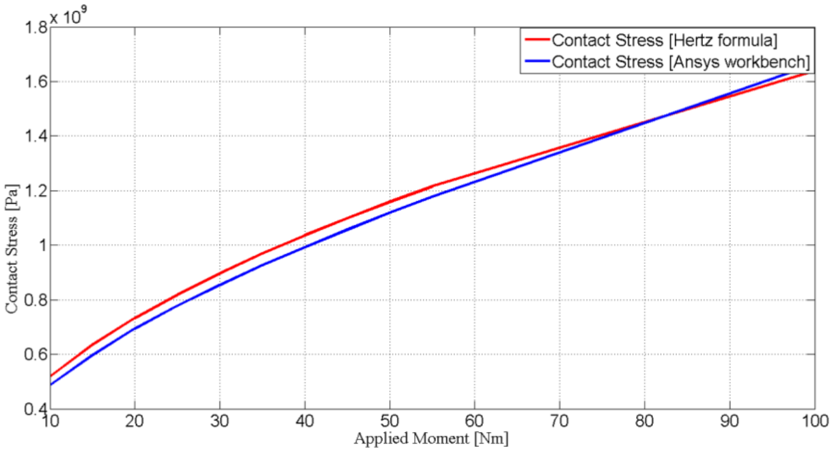

Fig. 5: Contact Stress Distribution of FEM and Hertz Equation with Error

The influence of coefficient of friction was also studied by taking five different values of coefficients as seen in Table 5 and Fig. 10 the contact stress increases with increase in coefficient of friction. Table 6 shows the graph of contact stress by Hertizean theory and by FEM for different Material at applied moment

CONCLUSION

In recent years the application of Finite Element Method to different applications is increasing taking advantage of powerful computational facilities. In this paper the contact stress in rolling-sliding contact of involute spur gear and the effect of coefficient of friction was analyzed. To achieve this, first, three dimensional involute spur gear pair was developed in Solid Works and the 3D model was exported to ANSYS software. Next, the analysis was done using both FEM and Hertizian stress formula. Thus, it is found that:

a) the contact stress both in the FEM and analytical method is the similar with minimum variation less than 6%;

b) the contact stress increases with higher coefficient of friction between the contact area; and

c) the applied analytical method verifies better performance of high strength materials.

REFERENCES

[1] ISO 6336 (1993), "*Calculation of Load Capacity of Spur and Helical Gears*".

[2] Ravichandra Patchigolla and Yesh P. Singh; "*Finite Element Analysis of Large Spur Gear Tooth and Rim With and Without Web Effects-Part I*"

[3] M. Raja Roy, S. Phani Kumar, D.S. Sai Ravi Kiran, "*Contact pressure analysis of spur gear using FEA*", International Journal of Advanced Engineering Applications, Vol.7, Iss.3, pp.27-41 (2014).

[4] Ali Raad Hassan Ali Raad Hassan , "*Contact Stress Analysis of Spur Gear Teeth Pair*", World Academy of Science, Engineering and Technology, Vol:3 2009 pp. 10-20

[5] Parveen Kumar, Harsh Raghuvanshi; "*Design & Analysis of a Spur Gear in different Geometric Conditions*", International Journal of Engineering and Advanced Technology (IJEAT),ISSN: 2249 – 8958, Volume-3, Issue-2, December 2013

[6] G. Mallesh,, Dr. V B Math, Gajanan, Uttesh, Sridhar, "*Estimation of Critical Section and Bending Stress Analysis for Asymmetric Spur Gear Tooth*", 14th National Conference on Machines and

Mechanisms (NaCoMM09), NIT, Durgapur, India, December 17-18, 2009.

[7] Muhsi n J. Jweeg & Wisam Auday Hussain, "*Contact Stresses for Different Gear Design Parameter*", Engineering and Technology journal , Vol 27, No.13, 2009.

[8] Daniel Tilahun REDDA, Tsutomu NAKANISHI and Gang DENG, Softening Behavior of Hardness and Surface Fatigue of Rolling-Sliding Contact in the Case of Developed Alloy Steels, Journal of Advanced Mechanical Design, Systems and Manufacturing, Vol. 3, No. 1, pp.85-92, 3, 2009.

[9] Zienkiewicz, O.C.; Taylor, R.L.; Zhu, J.Z; "*Finite element Method – its basics and fundamentals"*

[10] Dudley D.W., Handbook of Practical Gear Design, CRC Press, 1994.

[11] Gitin M Maitra, "*Handbook of gear design, 2nd edition"*, Tata McGraw-Hill Publishing company limited, 1994.

APPENDIX

Matlab Code for Hertizean Stress Analysis

```
%% Hertizean stress analysis
%% assume the radii of pinion and gear
are equal
%% assume the material of the gear and
pinion are same
% F is the force applied
%Rp1 and Rp2 are pitch radius of pinion
and gear respectively
%B is the face width
%phi is the pressure angle
%por1 and por2 is the poisson ratio of
pinion and gear respectively
%moe1 and moe2 are the modules of
elasticity of pinion and gear
%HCS= hertizean contact stress
%mom=moment applied in Nm
%HCS=sqrt[(F(1+(Rp1/Rp2)))/(Rp1*B*pi
*(((1-por1^2)/moe1)+((1-
por2^2)/moe2)*sin(phi))]
%%
closeall
clc
clearall
mo=input('Enter the moment applied :    ');
face_w=input('Enter the face width:    ');
poiss_rp=input('Enter the value of possion
ratio of pinoin:    ');
poiss_rg=input('Enter the value of possion
ratio of gear:    ');
Ymoe_p=input('Enter the value of youngs
modulus of elasticity of pinoin:');
Ymoe_g=input('Enter the value of youngs
modulus of elasticity of gear:');
pitch_rad_p=input('Enter the value of
pitch radius of pinion :');
pitch_rad_g=input('Enter the value of
pitch radius of gear :');
press_angle=input('Enter the value of
pressure angle in radian:   ');

mom=mo;
B=face_w;
Rp1=pitch_rad_p;
Rp2=pitch_rad_g;
F=mo/Rp1;
moe1=Ymoe_p;
moe2=Ymoe_g;
por1=poiss_rp;
por2=poiss_rg;
phi=press_angle;

%sample user input values are given
below:
% mo=50;
% B=7*10^-3;
% phi=(20/180)*pi;
% Rp1=0.033;
% Rp2=Rp1;
% Ymoe_p=206*10^9;
% Ymoe_g=Ymoe_p;
% poiss_rp=0.3;
% poiss_rg=poiss_rp;
%%
F = mom/Rp1;
HCS = sqrt((F*2)/((Rp1*B*pi)*2*((1-
por1^2)/(moe1*1))*sin(phi)));
```