International Journal of Engineering Research and Development
ISSN: 2278-067X, Volume 1, Issue 5 (June 2012), PP.38-43
www.ijerd.com

Influence of Friction on Contact Stress for Small Power Transmission Planetary Gear Box

Mr. J.R. Koisha[1], Mr. H.P. Doshi[2]
[1]*Post Graduate Student, Department Of Mechanical Engineering, L.D. College of Engineering, Ahmedabad, Gujarat.*
[2]*Associated Professor, Department Of Mechanical Engineering, L.D. College of Engineering, Ahmedabad, Gujarat.*

Abstract—In planetary gear box, load is shared by planets, therefore to know the behaviour of contact is significant in it. In the present study parametric solid model of planetary gear box with three planets having involute profiled spur gears is modelled by Pro-Engineer Wildfire 5.0 software. Contact stress analysis of the three dimensional solid model of the planetary gear box requires huge computational resources, therefore plane stress frictionless contact analysis of the planetary gear box is carried out in ANSYS Workbench 12.0 and the results are verified by theoretical calculation of contact stress as per ISO 6336 standard. Then plane stress frictional contact analysis of the planetary gear box is done with the different values of coefficient of friction. The relationship of contact stress and coefficient of friction has showing a linear relationship. Further it is observed that at the low power transmission the slope of the graph is less.

Keywords—planetary gear box, contact analysis, FEA, frictional contact, plane stress analysis

I. INTRODUCTION

During the recent past, significant progress in the field of contact analysis of gears has also been made, and finite element analysis (FEA) is gradually becoming established as an efficient tool in gear box design. Using the finite element analysis, which is a general and systematic computational procedure for approximately solving problems in physics and engineering, many contact problems, ranging from relatively simple ones to quite complicated ones, can be solved with high accuracy. The Finite Element Method can be considered the favourite method to treat contact problems, because of its proven success in treating a wide range of engineering problem in areas of solid mechanics, fluid flow, heat transfer, and for electromagnetic field and coupled field problems.

II. CONTACT ANALYSIS OF GEARS

The first works of development of Tooth Contact Analysis have been done by Litvin and Kai, and Baxter. Significant contributions to the development of Tooth Contact Analysis have been made by the engineers of the Gleason Works, Klingelnberg, and Oerlikon [1]. O. Vogel et al, (2002) presented a constructive approach for the approximation free tooth contact analysis of hypoid bevel gears [2]. A.R. Mijar and J.S. Arora (2004) developed an augmented Lagrangian optimization method and discussed for contact analysis problems that automatically update the user specified penalty values to obtain the final appropriate values. Further, to solve the frictional contact analysis problem accurately, a two phase formulation is proposed [3]. Shuting Li (2005) presented three-dimensional, finite element methods to conduct surface contact stress and root bending stress calculations of a pair of spur gears with machining errors, assembly errors and tooth modifications [4]. In the present study attempt is made to estimate effect of friction on contact stress in planetary gear box using non-linear FEA.

III. PROBLEM FORMULATION

The selected specification of the planetary gear box is as follow:

TABLE I
SPECIFICATION OF THE PLANETARY GEAR BOX

Sr. no.	Stage	Gear	teeth	Dia.	Torque in Nm
1	Stage 1	Sun	20	30	-28.64
		Pinion	55	82.5	--
		Annulus	130	195	214.84
		Carrier	--	--	-186.20
2	Stage 2	Sun	20	60	214.84
		Pinion	64	192	--
		Annulus	148	444	-1804.72
		Carrier	--	--	-1589.87

Power, P = 1500 W, Input Speed $N_{input} = 500$ r.p.m., Output Speed $N_{output} = 7.9365$ r.p.m., Reduction Ratio, i = 63:1, Two Stages, i_1=7.5, i_2=8.4, Gear Material = 16MnCr5

IV. CONTACT STRESS CALCULATION USING ISO 6336

The Standard ISO 6336 Part2 (1996a) describes the following contact stress equation based on a different defined set of factors:

$$\sigma_C = Z_H Z_E Z_\varepsilon \sqrt{\frac{F_t}{bd_S} \cdot \frac{(i+1)}{i} K_A K_V K_{H\alpha} K_{H\beta}}$$

Where,
Z_H = it is the zone factor, which takes into account the flank curvatures at the pitch point and transforms tangential load at the reference cylinder to tangential load at the pitch cylinder.
Z_E = it is the elasticity factor, which takes into account specific properties of the material.
$Z\varepsilon$ = it is the contact ratio factor, which takes into account the influence of the effective length of the lines of contact.
K_A = it is the application factor, which takes into account the load increment due to externally influenced variations of input or output torque.
K_V = it is the dynamic factor, which takes into account load increments due to internal dynamic effects.
$K_{H\alpha}$ = it is the transverse load factor for contact stress, which takes into account uneven load distribution in the transverse direction resulting, for example, from pitch deviation.
$K_{H\beta}$ = it is the face load factor for contact stress, which takes into account uneven distribution of load over the face width, due to mesh misalignment caused by inaccuracies in manufacture, elastic deformations, etc.

A. For the first stage:

$$\sigma_C = Z_H Z_E Z_\varepsilon \sqrt{\frac{F_t}{bd_S} \cdot \frac{(i_1+1)}{i_1} K_A K_V K_{H\alpha} K_{H\beta}}$$

Where,
Z_H = 1.5 ($\because$ ISO 6336-2, clause-6 Graph)
Z_E = 189.8 $\frac{N}{mm^2}$ ($\because$ ISO 6336-2, clause-7 Table-1 for steel)
$Z\varepsilon$ = 0.7 ($\because$ ISO 6336-2, clause-8 from fig-4)
K_A = 1 ($\because$ ISO 6336-1, clause-5)
K_V = 1 ($\because$ ISO 6336-2, clause-6, 6.1)
$K_{H\alpha}$ = 1.5 ($\because$ ISO 6336-1, Fig-16; Graph)
$K_{H\beta}$ = 1 ($\because$ ISO 6336-6, Table C-2)

$$\therefore \sigma_C = 309.05 \ \frac{N}{mm^2}$$

B. For the second stage:

$$\sigma_C = Z_H Z_E Z_\varepsilon \sqrt{\frac{F_t}{bd_S} \cdot \frac{(i_2+1)}{i_2} K_A K_V K_{H\alpha} K_{H\beta}}$$

Where,
Z_H = 1.5 ($\because$ ISO 6336-2, clause-6 Graph)
Z_E = 189.8 $\frac{N}{mm^2}$ ($\because$ ISO 6336-2, clause-7 Table-1 for steel)
$Z\varepsilon$ = 0.65 ($\because$ ISO 6336-2, clause-8 from fig-4)
K_A = 1 ($\because$ ISO 6336-1, clause-5)
K_V = 1 ($\because$ ISO 6336-2, clause-6, 6.1)
$K_{H\alpha}$ = 1.5 ($\because$ ISO 6336-1, Fig-16; Graph)
$K_{H\beta}$ = 1 ($\because$ ISO 6336-6, Table C-2)

$$\therefore \sigma_C = 276.14 \ \frac{N}{mm^2}$$

V. PARAMETRIC SOLID MODELLING OF THE PLANETARY GEAR BOX

Parametric means that the physical shape of the part or assembly is driven by the values assigned to the attributes (primarily dimensions) of its features. The parametric solid models of involute 20° full depth profiled gears are generated in Pro-Engineer Wildfire 5.0. and then assembled in the same. This model of the planetary gear box is transferred in ANSYS Workbench 12.0.

VI. TEN STEPS OF FEA IN ANSYS WORKBENCH 12.0

1. In this present study, geometry is imported from Pro-Engineer Wildfire 5.0 modelling software.
2. Choose proper type of analysis i.e., structural, thermal, etc. Frictional contact stress analysis is chosen. It is a type of static structural analysis. The behaviour of the stresses in planetary gear box having spur gears is plane stress. One can choose plane stress analysis when the normal stress and the shear stresses directed

perpendicular to the plane are assumed to be zero. The main advantage of plane stress analysis is it requires very less computational resources for the same domain. In this problem Plane stress analysis can be selected with the facts: (i) the geometry of spur gear along the axis of rotation is uniform and there is absence of axial loading. (ii) the assembly error is not modelled in this study. (iii) the stresses generated in carrier are ignored because the here focus of the study is contact stress generated at teeth meshing.

3. Choose proper unit system which will be used to define various properties.
4. Define material properties which are necessary to solve the problem. Here the gear material is 16MnCr5 and the required material properties for this analysis are only Modulus of elasticity, Poison's Ratio and Density. The Modulus of elasticity of 16MnCr5 is 206 GPa. The Poison's Ratio of 16MnCr5 is 0.3. The Density of 16MnCr5 is 7870 kg/m3.
5. Generate new coordinate system if required.
6. Define contact type & formulation method of contact, if the problem has contact nonlinearity. The type of contact determines how the contacting bodies can move relative to one another. The types of connections available in ANSYS Workbench are Bonded, No separation, Frictionless, Rough and Frictional. Choosing the appropriate contact type depends on the type of problem one is trying to solve. If the stresses very nearer to a contact interface are important, use one of the nonlinear contact types. Here, in this analysis frictionless and frictional contact types are used. However, use of nonlinear contact types usually results in longer solution times and can have possible convergence problems due to the contact nonlinearity. Here, Augmented Lagrange formulation method is selected. It is best suitable method among available methods for the frictional contact type which is nonlinear type analysis.

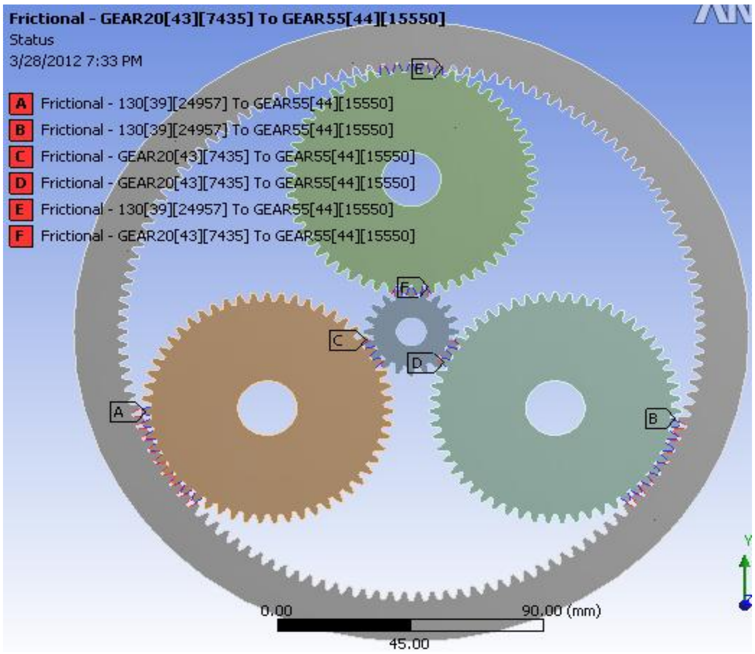

Fig.1 the contacts defined

7. Generate appropriate mesh using various meshing control parameters.

Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	On: Proximity and Curvature
Relevance Center	Medium
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Proximity Accuracy	0.5
Num Cells Across Gap	Default (3)
Min Size	Default (8.0656e-002 mm)
Max Face Size	Default (8.06560 mm)
Max Tet Size	Default (16.1310 mm)
Growth Rate	Default (1.20)
Minimum Edge Length	0.315210 mm

Fig.2 Mesh Generation options

Mesh should be enough fine at the area of interest (location in the part where stresses are critical) and mesh should be enough coarse at the remaining areas. This combination of fine and coarse mesh gives accurate results with optimum use of computational resources. The figure 2 shows set of meshing option parameters used. The figure 3 shows generated mesh.

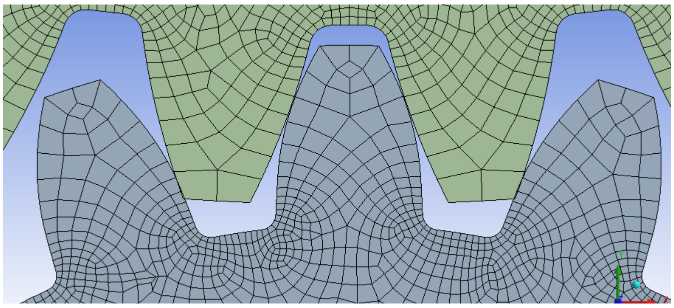

Fig.3 Generated Auto Mesh

8. Apply boundary conditions which really represent actual loading conditions. Boundary condition refers to the external load on the border of the structure. If one makes any mistake in simulation of actual boundary conditions, software will give wrong results. Therefore it is very critical to decide the boundary conditions in software. Here the four boundary conditions are applied on the geometry (shown in figure 4), (i) frictionless support at the sun gear centre. It means that sun gear is free to rotate on its axis. (ii) fixed support at the outer periphery of the annulus gear. It means that annulus gear is fixed. (iii) frictionless support at the planet gears centers. It means that planet gears are free to rotate on its axis but carrier arm is fixed, which generates resistance to the torque. (iv) moment is applied on the sun gear. It means that gearbox is transmitting power.

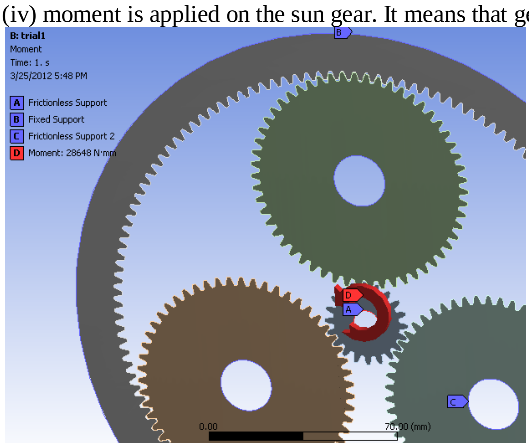

Fig.4 boundary conditions

9. Solve the FEA model using proper solver.
10. Analyse the results using various displays.

VII. RESULTS

The study was undertaken to investigate the effect of coefficient of friction on contact stress. First analysis is done without friction means with frictionless contact and that result is compared with ISO, which validates the results of ANSYS Workbench. Here in the table II the results obtained by ISO standard and ANSYS Workbench are compared.

TABLE III: COMPARISON FRICTIONLESS CONTACT STRESS

Sr. No.	Method	σ_C as per ISO standard in N/mm^2	σ_C as per ANSYS Workbench in N/mm^2	% variation
1	stage 1	309.05	318.25	2.89 %
2	stage 2	276.14	275.16	-0.356 %

The stress calculated by ANSYS Workbench match with the stress calculated by ISO standard. The variation is up to 3%. This shows that ANSYS Workbench gives accurate result and one can use the same procedure to calculate frictional contact stress by ANSYS Workbench. The stress pattern at different place is shown in figure 5.

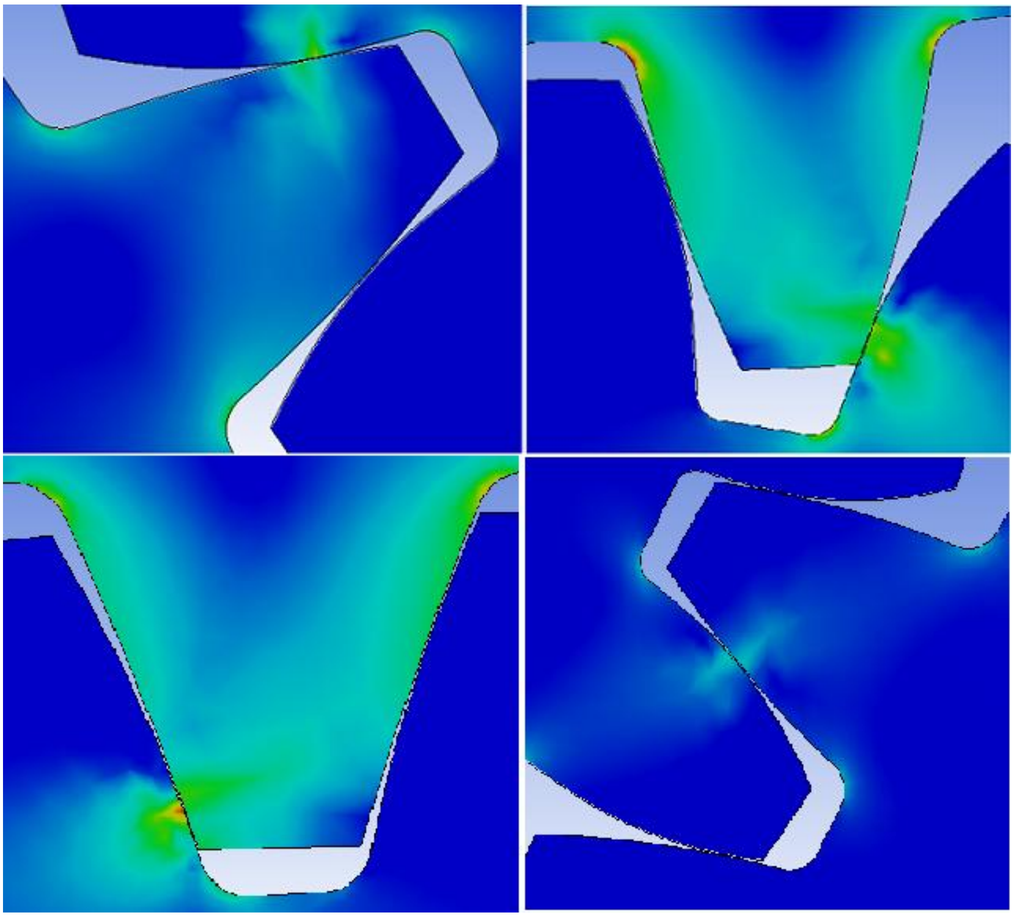

Fig.5 stress pattern

Frictional contact analysis is done with different values of coefficient of friction ranging from 0.05 to 0.15 and the results are shown in table III.

TABLE IIIII : RESULTS OF FRICTIONAL CONTACT ANALYSIS

stage 1		**stage2**	
μ	**contact stress**	**μ**	**contact stress**
0	318.25	0	275.16
0.05	322.31	0.05	279.13
0.065	325.35	0.065	279.52
0.07	326.10	0.08	280.20
0.085	325.97	0.09	280.92
0.1	327.80	0.1	282.48
0.115	329.62	0.115	281.67
0.13	331.51	0.13	283.20
0.15	334.11	0.15	283.08

The trend lines of the above results are plotted in excel sheet which are shown in figure 6 and 7 for both stages of gearbox.

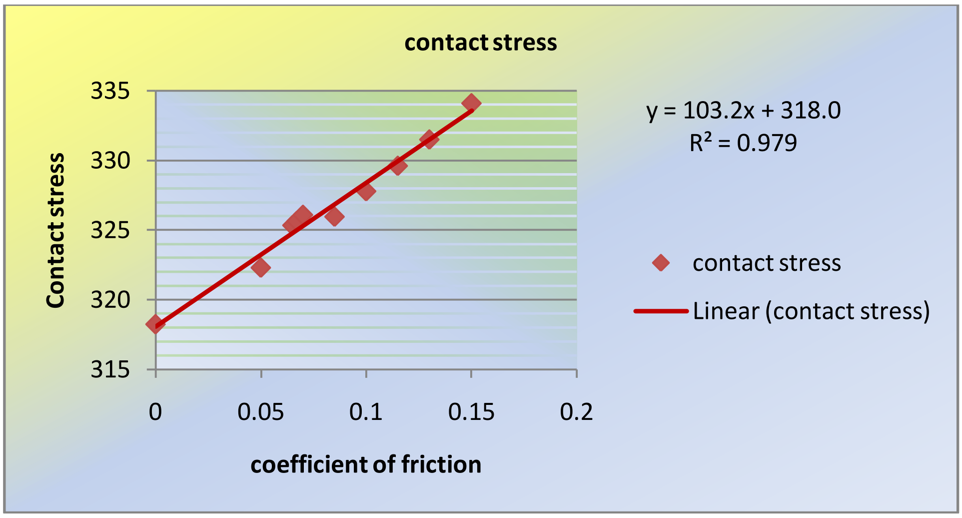

Fig.6 effect of coefficient of friction on contact stress of stage 1

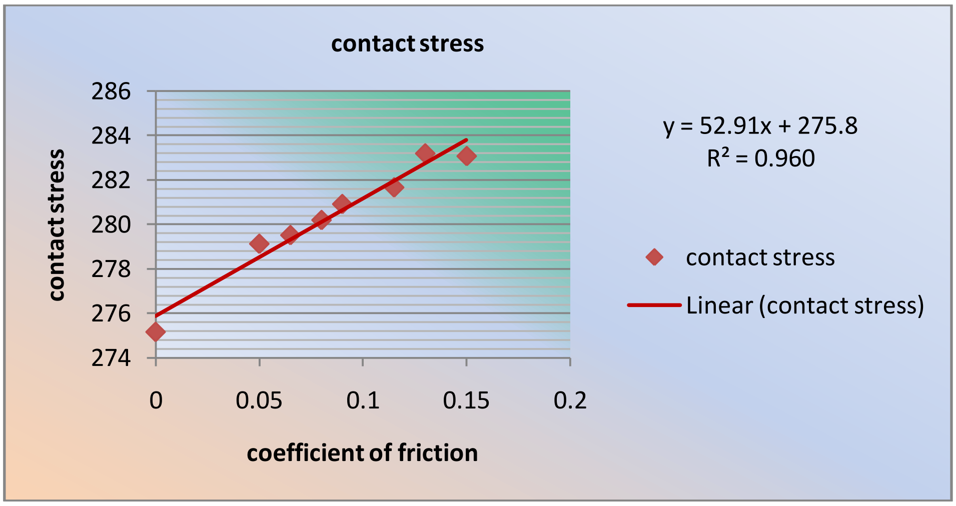

Fig.7 effect of coefficient of friction on contact stress of stage 2

VIII. CONCLUSION

R^2 value on chart is called correlation coefficient or coefficient of determination and it shows that the reliability of the trend and accuracy of the forecast which is greater than 0.95. Therefore trend line shows that frictional contact stress follows linear trend with coefficient of friction and with the help of it we can find value of frictional contact stress for different values of coefficient of friction for the particular stage of gearbox. The slope of the graph is less for low power transmission planetary gear box.

REFERENCES

[1]. Faydor L. Litvin [a], Galina I. Sheveleva [b], Daniele Vecchiato [a], Ignacio Gonzalez-Perez [c], Alfonso Fuentes [c], *Modified approach for tooth contact analysis of gear drives and automatic determination of guess values*, [a] Gear Research Center, Department of Mechanical and Industrial Engineering, University of Illinois at Chicago, [b] Department of Theoretical Mechanics, Moscow State University of Technology 3A, Vadkovsky per., Moscow, 101472, Russia, [c] Polytechnic University of Cartagena, Department of Mechanical Engineering, Campus Universitario Muralla del Mar, C/Doctor Fleming, Cartagena 30202, Spain, 2004.

[2]. O. Vogel [a], A. Griewank [a], G. Bar [b], *Direct gear tooth contact analysis for hypoid bevel gears*, [a] Institute of Scientific Computing, Technical University of Dresden, 01062 Dresden, Germany, [b] Institute of Geometry, Technical University Dresden, 01062 Dresden, Germany, 2002.

[3]. A.R. Mijar and J.S. Arora, *An augmented Lagrangian optimization method for contact analysis problems, 1: formulation and algorithm*, Optimal Design Laboratory, College of Engineering, The University of Iowa, Iowa City, IA 52242, USA, 2004.

[4]. Shuting Li, Finite element analyses for contact strength and bending strength of a pair of spur gears with machining errors, assembly errors and tooth modifications, Nabtesco Co. Ltd. Oak-hills No. 202, Heki-cho 7028-2, TSU-shi, Mie-ken 514-1138, Japan, 2005.