International Journal of Modern Studies in Mechanical Engineering (IJMSME)
Volume 1, Issue 1, June 2015, PP 45-52
www.arcjournals.org

Contact Stress Analysis of Spur Gear for Different Materials using ANSYS and Hertz Equation

Putti Srinivasa Rao[1], **Nadipalli Sriraj**[2], **Mohammad Farookh**[2]

[1]Professor, Dept. of Mechanical Engineering, Andhra University College of Engineering (A),
Visakhapatnam, Andhra Pradesh, India.
s_putti@rediff.com
[2]B.E Project Students, Dept. of Mechanical Engineering, Andhra University College of Engineering (A), Visakhapatnam, Andhra Pradesh, India.

Abstract: *The contact stress in the mating gears is the key parameter in gear design. Deformation of the gear is also another key parameter which is to be considered. Gears generally fail when the working stress exceeds the maximum stress. The study in this paper shows that the complex design problem of spur gear which requires fine software skill for modeling and also for analyzing. The project aims at the minimization of both contact stress as well as deformation to arrive at the best possible combination of driver and driven gear. In this process of spur gears mating, 3 different materials were selected and the software programme was performed for 9 different combinations to get the best result possible. The results of the two dimensional FEM analysis from ANSYS are presented. These stresses were compared with the theoretical Hertz's equation values. Both results agree very well. This indicates that the FEM model is accurate.*

Keywords: *Contact stress, Spur gear, Hertz's equation, ANSYS, FEM.*

1. INTRODUCTION

Gears are one of the oldest of humanity's inventions. Nearly all the devices we think of as machines utilize gearing of one type or another. . Nevertheless, the design or specification of a gear is only part of the overall system design picture. From industry's standpoint, gear transmission systems are considered one of the critical aspects of Contact Stress Analysis. Investigators analyzing the gear tooth for stresses have done several studies. Research article on Modeling and Finite Element Analysis of Spur Gear is done by Vivek Karaveer, Ashish Mogrekar and T. Preman Reynold Joseph [1] in which the contact stresses of gears is found out by ANSYS and HERTZ equation which is for finding contact stress in gears. In detail study of the contact stress produced in the mating gears is the most important task in design of gears as it is the deciding parameter in finding the dimensions of gear (Mr. Bharat Gupta et al 2012)[2]. DhavaleA.S., Abhay Utpat [3] paper explores when gear is subjected to load, high stresses developed at the root of the teeth, Due to these high stresses, possibility of fatigue failure at the root of teeth of spur gear increases. T. Shoba Rani et al. [4] have used cast iron, nylon and polycarbonate as the materials used for the project of spur gear for finite element analysis. Sushil Kumar Tiwari [5] found out the contact stress and bending stress for involute spur gear teeth in meshing by finite element method.

2. THEORETICAL CALCULATION OF CONTACT STRESSES

PITTING is a surface fatigue failure resulting from repetitions of high contact stress. The surface fatigue mechanism is not definitively understood. Pitting commonly appears on operating surfaces of gear teeth, a fundamental cause is excessive loading that raised contact stresses beyond critical levels. Contact stress has been expressed clearly in this work by finite element model; according to the angular motion. The domain in this study was the angular location of the point of contact on the arc of action from the beginning to the end of contact between this pair of teeth. Target and contact elements were used in both sides of the contact pattern between the surfaces of this pair of teeth.

2.1 Spur Gear Contact

The transfer of power between gears takes place at the contact between the acting teeth. The stresses at the contact point are computed by means of the theory of Hertz. The theory provides mathematical expressions of stresses and deformations of curved bodies in contact. Fig. 1 shows a model applied to the gear-two parallel cylinders in contact.

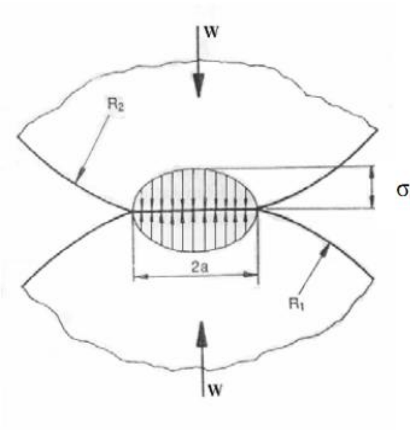


Fig1.*Parallel cylinders in contact*

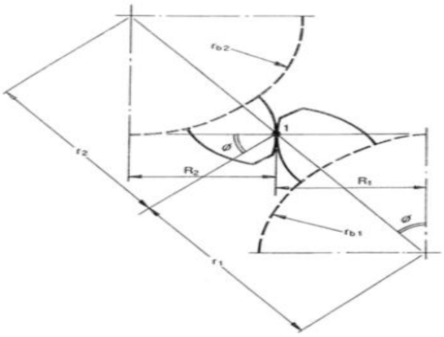


Fig2.*Two involutes teeth in contact*

The Hertz theory assumes an elliptic stress distribution, as seen in the Fig. 1; the maximum stress

$$\sigma_o = \sqrt{\frac{W(1/R_1 + 1/R_2)}{F\pi[(1-\nu_1^2)/E_1 + (1-\nu_2^2)/E_2]}} \tag{2}$$

σ_c= maximum value of contact stress (N/mm^2)

W= force pressing the two cylinders together (N)

B = half width of deformation (mm)

L= axial length of cylinders (mm)

d1, d2 = diameters of two cylinders (mm)

E1, E2 = modulii of elasticity of two cylinder materials (N/mm^2)

µ1, µ2= Poisson's ratio of the two cylinder materials (Unit less)

Where W is the load, F is the face width of pinion. Same equation can be apply for teeth, assuming for R1 and R2 the respective radii of the involute curve at the contact point, as shown in Figure. Let us assume that the contact takes place at point 1, and then the respective radii are equal to: $R_1 = r_{p1}\sin\varphi$; $R2 = r_{p2}\sin\varphi$

Hence, the Hertz equation for contact stresses in the teeth becomes,

$$\sigma_o = \sqrt{\frac{W(1 + r_{p1}/r_{p2})}{r_{p1}F\pi[(1-\nu_1^2)/E_1 + (1-\nu_2^2)/E_2]\sin\phi}} \tag{3}$$

Where rp1 and rp2 are the pitch radii of the pinion and gear and φ is the pressure angle. The stress correlations derived heretofore and Eq. (3) are based on a number of simplifying assumptions, such as pure bending of short beam and elliptic distribution of stresses at tooth contact.

3. DETERMINATION OF CONTACT STRESS IN ANSYS

ANSYS originally a finite element analysis (FEA) code for structural mechanics. ANSYS Workbench contains Design Modeler in which the gear is designed from basic drawing tools.

3.1 Design of Spur Gear in ANSYS Design Modeler

The model is done in design modeler of ANSYS Workbench. In this study, maximum contact stress is determined. During the transmission of torque of 15000 lb-in or 1694.7725 Nm by steel, grey cast iron, aluminum spur gears, using finite element analysis. The dimensions of the gears are given in Table below.

Table1.*Spur Gear Specifications*

Dimension	Unit	Symbol	Value (For both gears in assembly)
Number of Teeth	-	Z	20
Pitch Circle Diameter	mm	D	127
Pressure Angle	Degrees	Φ	20
Addendum Radius	mm	R_A	69.85
Dedendum Radius	mm	R_B	55.88
Face Width	mm	B	101.60
Shaft Radius	mm	R_s	31.75

ANSYS Design Modeler gives you some handful of options in order to create a working design of the spur gear. The spur gear is designed by giving the coordinates of the involute curve from involute equation and creating the profile of an involute. Then the basic operations like replicate are used to duplicate the involute curve and reverse it in order to get the profile of gear.

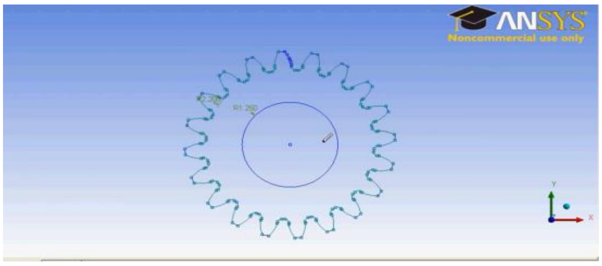


Fig3.*spur gear design in Design Modeler*

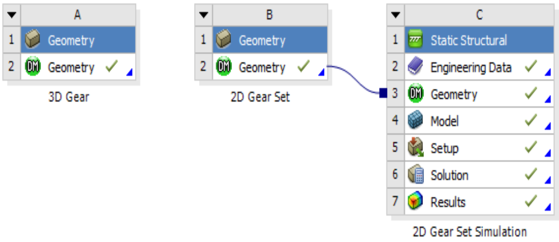


Fig4.*static structural*

The materials comprises of structural steel, grey cast iron aluminum. First two materials are conventional materials whereas the other two materials are a kind of advanced materials to deal with the contact stresses. The material properties are entered in ANSYS Engineering data and the formulation is completed.

Table2. *Spur Gear material specifications*

MATERIAL	STRUCTURAL STEEL	GREY CAST IRON	ALUMINIUM
Young’s Modulus(Mpa)	2E+5	1.1E+5	0.7E+5
Poisson’s Ratio	0.3	0.28	0.33

3.2 Automatic Mesh Generation

Mesh generation is one of the most critical aspects of engineering simulation.. ANSYS [6] Meshing technology provides a means to balance these requirements and obtain the right mesh for each simulation in the most automated way possible.

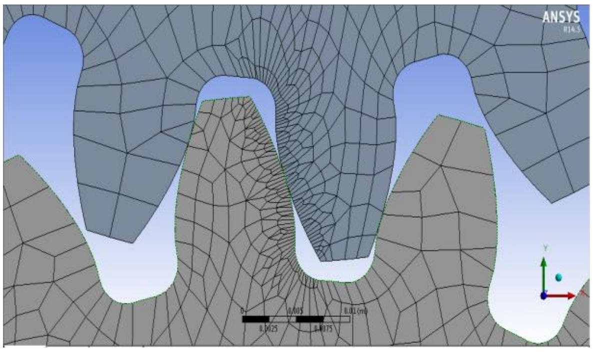


Fig5.*Mesh Generation*

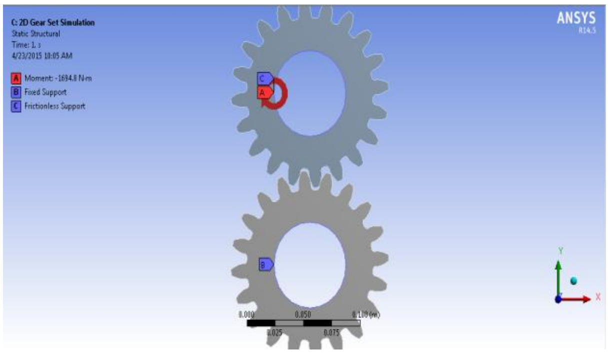


Fig6.*2D Gear Set Simulation*

3.3 Stress Analysis in ANSYS Mechanical

Fixed support is applied on inner rim of the lower gear. Frictionless support is applied on the inner rim of upper gear to allow its tangential rotation but restrict from radial translation. Moment of 15000 lb-in or 1694.7725 N-m is applied on the inner rim of upper gear in clockwise direction as a driving torque. The mating gear is created by translating a copy of the original copy of gear and by giving an interference which is equal to the center distance of the two mating gears.

3.4 ANSYS Results

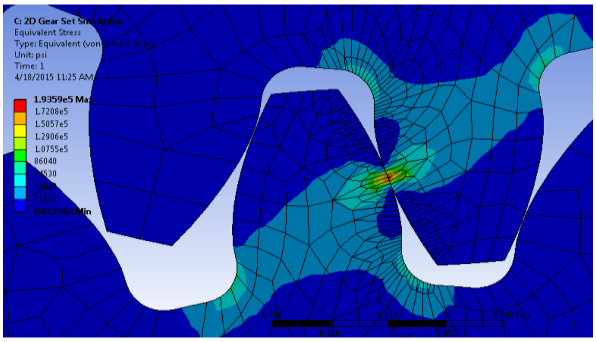


Fig7(a).*Stress b/w Steel (driver) & steel (driven)*

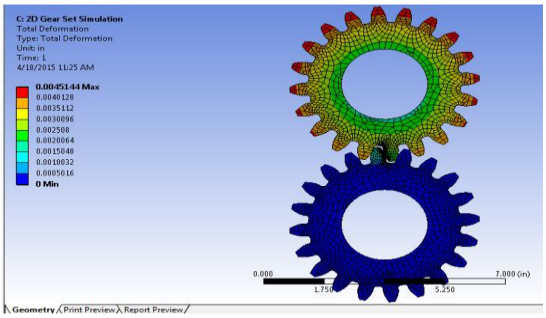
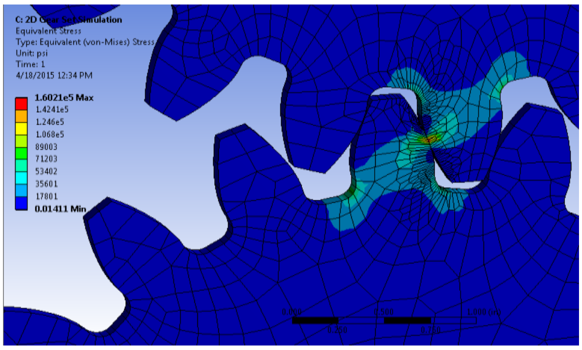


Fig7(b).*Deformation b/w steel (driver) & steel (driven)*



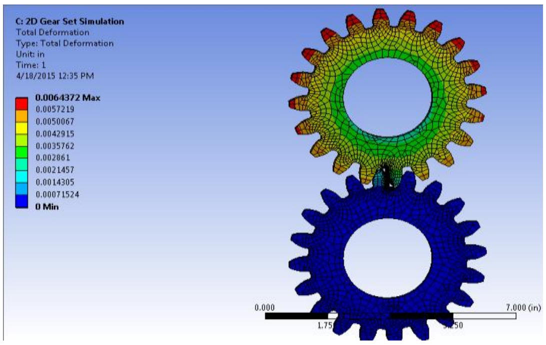


Fig8(a).*Stress b/w Steel (driver) & cast iron (driven)*

Fig8(b).*Deformation b/w steel (driver) & cast iron (driven)*

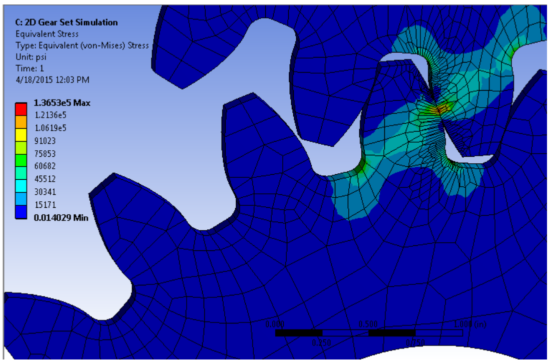


Fig.9 (a). *Stress b/w Steel (driver) & aluminium (driven)*

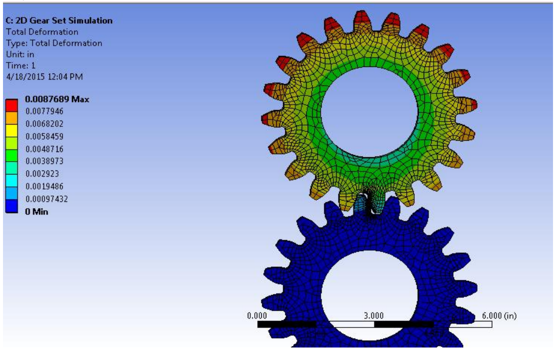


Fig.9 (b). *Deformation b/w steel (driver) & aluminium (driven)*

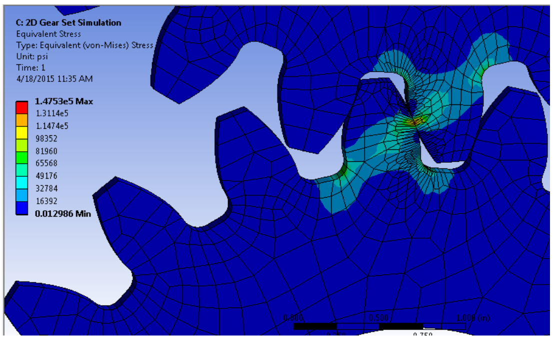


Fig.10 (a) *Stress b/w Cast iron(driver) & steel(driven)*

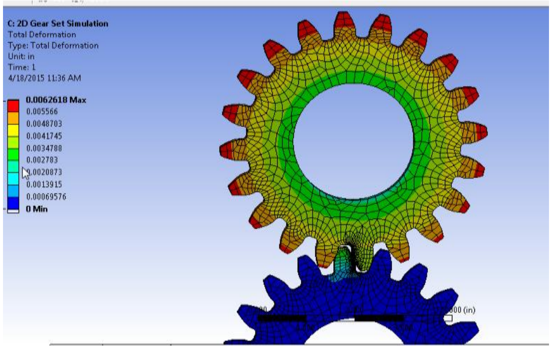
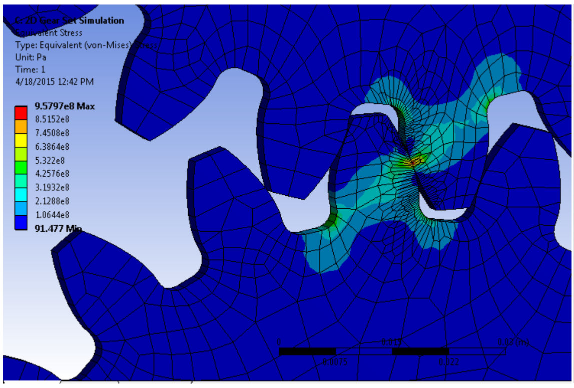


Fig.10(b) *Deformation b/w cast iron(driver) & steel(driven)*



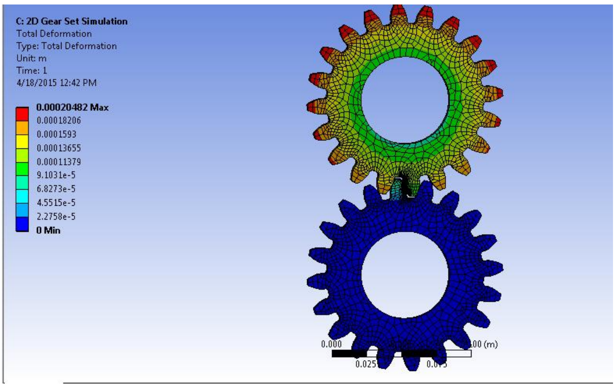
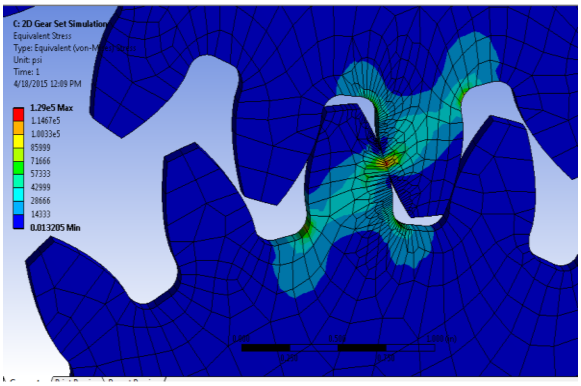


Fig.11(a). *Stress b/w Cast iron(driver) & cast iron(driven)* **Fig.11(b)** *.Deformation b/w cast iron(driver) & cast iron (driven)*



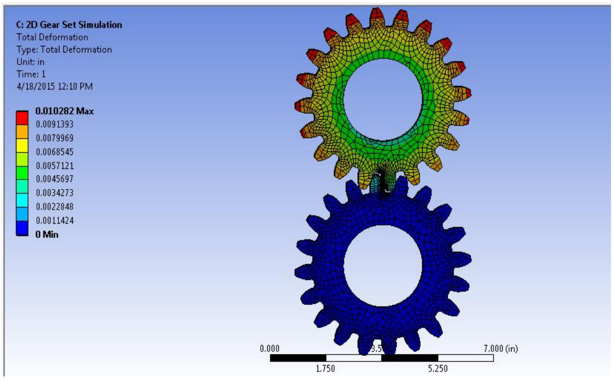
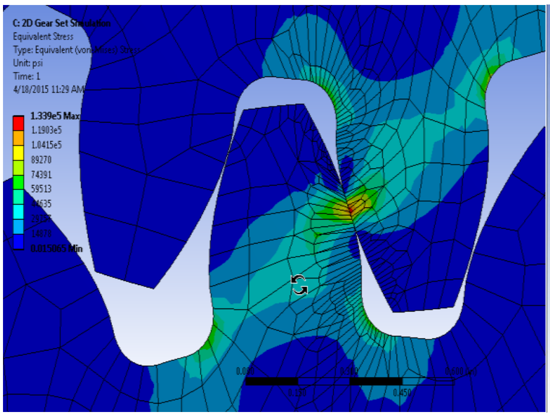


Fig. 12(a) *Stress b/w Cast iron(driver) & aluminum(driven)* **Fig.12 (b)** *Deformation b/w cast iron (driver) & Aluminum (driven)*



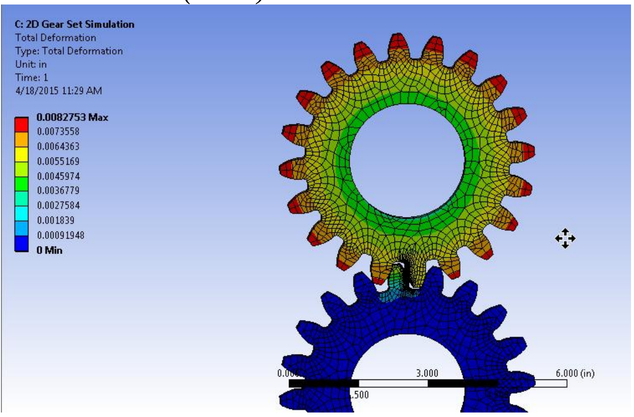
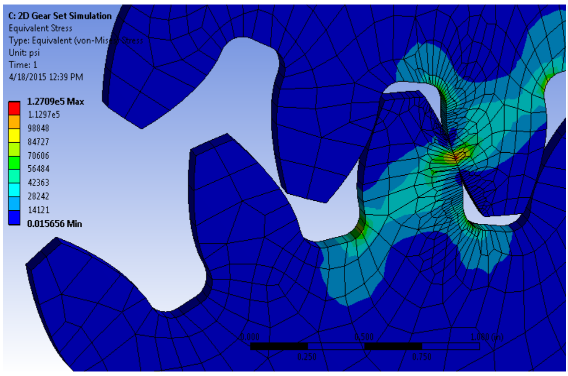


Fig.13 (a) *Stress b/w Aluminum (driver) & steel (driven)* **Fig. 13(b)** *Deformation b/w aluminum (driver) &steel (driven)*



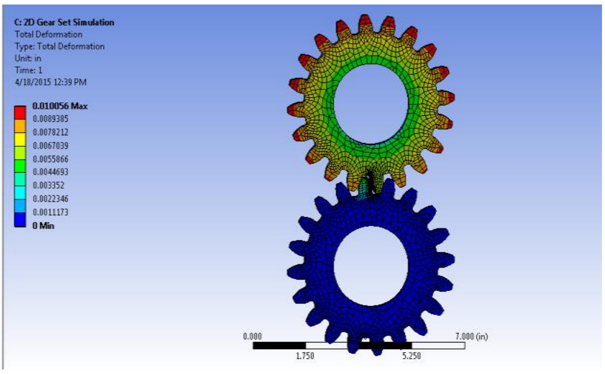


Fig.14 (a) *Stress b/w aluminium(driver) & cast iron(driven)* **Fig.14(b)** *Deformation b/w aluminium(driver) & cast iron (driven)*

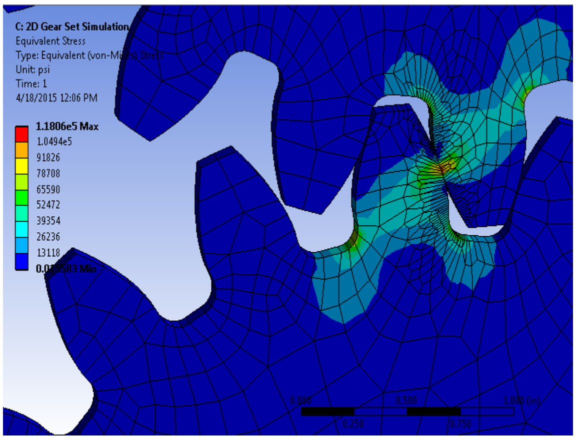


Fig.15 (a) *Stress b/w aluminium (driver) & aluminum (driven)*

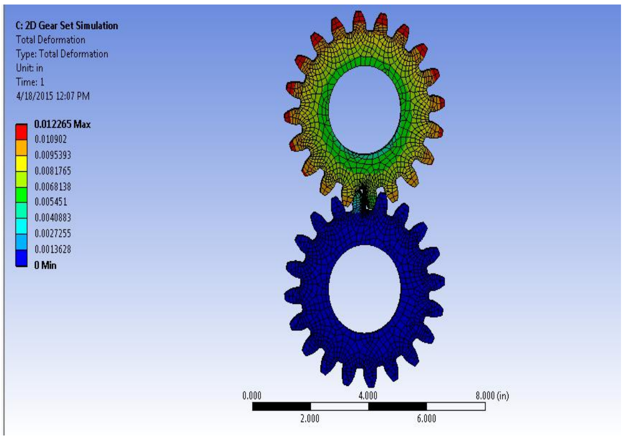


Fig.15 (b) *Deformation b/w aluminum (driver) & aluminium (driven)*

Observation of ANSYS results:

Fig. 7(a) shows the contact stress between steel (driver) and steel (driven) = 1334.76 Mpa

Fig. 7(b) shows the deformation between steel (driver) and steel (driven) = 0.0045144 m

Fig. 8(a) shows the contact stress between steel (driver) and cast iron (driven) = 1104.54 Mpa

Fig. 8(b) shows the deformation between steel (driver) and cast iron (driven) = 0.0664372 m

Fig.9 (a) shows the contact stress between steel (driver) and aluminium (driven) = 941.34Mpa

Fig. 9(b) shows the deformation between steel (driver) and aluminium (driven) = 0.0087689 m

Fig. 10(a) shows the contact stress between cast iron (driver) and steel (driven) = 1017.18 Mpa

Fig. 10(b) shows the deformation between cast iron (driver) and steel (driven) = 0.0062618 m

Fig. 11(a) shows the contact stress between cast iron(driver) and cast iron (driven) = 889.42 Mpa

Fig. 11(b) shows the deformation between cast iron (driver) and cast iron (driven) = 0.0002048 m

Fig. 12(a) shows the contact stress between cast iron (driver) and aluminum (driven) = 889.42 Mpa

Fig. 12(b) shows the deformation between cast iron (driver) and aluminium (driven) = 0.0120282 m

Fig. 13(a) shows the contact stress between aluminum (driver) and steel (driven) = 923.2 Mpa

Fig. 13(b) shows the deformation between aluminum (driver) and steel (driven) = 0.0082753 m

Fig. 14(a) shows the contact stress between aluminum (driver) and cast iron (driven) = 876.25 Mpa

Fig. 14(b) shows the deformation between aluminium (driver) and cast iron (driven) = 0.010056 m

Fig. 15(a) shows the contact stress between aluminum (driver) and aluminium (driven) = 813.99 Mpa

Fig. 15(b) shows the deformation between aluminium (driver) and aluminium (driven) = 0.012265 m

4. MODEL CALCULATIONS

4.1 Calculation of Contact Stresses Using ANSYS

The maximum Contact stress using ANSYS is 1334.756 MPa

The maximum Deformation accounts for 0.000111467 Meters

4.2 Theoretical Calculation of Contact Stresses by Hertz Equation

Torque = Force (F) * Shaft Radius (R_s)

1694.7725e3 (N-mm) = F * 31.75 (mm)

F = 53378.6614 N

$$\sigma_c = \sqrt{\frac{F(1+R_1/R_2)}{R_1 B\pi(\frac{1-\mu_1^2}{E_1}+\frac{1-\mu_2^2}{E_2})\sin\phi}}$$

$$= \sqrt{\frac{53378.6614(1+63.5/63.5)}{63.5 * 101.6 * \pi(\frac{1-0.3^2}{2*10^5}+\frac{1-0.3^2}{2*10^5})\sin 20}}$$

= 1300.899 MPa

Contact stress from hertz equation is 1300.899 MPa

% ERROR in contact stress of ANSYS from HERTZ formula

$$= \frac{(1334.756-1300.899)*100}{1300.899} = 2.602\%$$

5. RESULTS AND CONCLUSION

The use of different materials in gear manufacturing provides a range of contact stresses. This range of contact stresses and deformation is useful in the selection of material in different applications. The use of different materials in gear manufacturing provides a range of contact stresses. This range of contact stresses and deformation is useful in the selection of material in different applications. The values obtained by Hertz's equation and Ansys agree with each other with each other with a maximum error of 4% which is acceptable. It shows that the simulation done in ANSYS is compatible and copes up with the hertz equation for a range of materials used in the experiment. Aluminum on the other hand provides less contact stress when mated with any of the gears. The lowest contact stress is recorded when aluminum is used as both driver as well as driven gear. The Contact stresses from ANSYS and Hertz equation are tabulated as below.

Table.3 *Contact stress result*

MATERIALS	CONTACT STRESS (Mpa)		
	ANSYS	Hertz	Error
Structural Steel & Structural Steel	1334.76	1300.89	2.603
Cast Iron & Structural Steel	1017.18	1091.4	6.8004
Aluminium & Structural Steel	923.2	948.98	2.792
Structural Steel & Cast Iron	1104.54	1091.4	1.2
Cast Iron & Cast Iron	889.42	857.8	3.73
Aluminium & Cast Iron	876.25	919.83	4.97
Structural Steel & Aluminium	941.34	948.98	0.805
Cast Iron & Aluminium	889.42	919.83	3.419
Aluminium & Aluminium	813.99	783.27	3.774

REFERENCES

[1]. VivekKaraveer, AshishMogrekar and T. PremanReynold Joseph "Modeling and Finite Element Analysis of Spur Gear" International Journal of Current Engineering and Technology Vol.3, No.5, December-2013.

[2]. Gupta, Mr Bharat, "Contact stress analysis of spur gear", In International Journal of Engineering Research and Technology, Vol. 1, No. 4, ESRSA Publications, June-2012.

[3]. Dhavale A.S and AbhayUtpat, "Gear tooth stresses using ANSYS" International Journal of Current Engineering and Technology Vol.2, December-2012.

[4]. T.Shoba Rani, "SPUR GEAR."International Journal of Computational Engineering Research", Vol. 3, No.11, 2013.

[5]. Sushil Kumar Tiwari, "Stress analysis of mating involute spur gear teeth", In International Journal of Engineering Research and Technology, Vol.1, No. 9, ESRSA Publications, November-2012.

[6]. Putti Srinivasa Rao and Ch.Ratnam, “Experimental and Analytical modal analysis of welded structures used for vibration based damage identification”, Global Journal of Researches in Engineering: Mechanical and Mechanics Engineering, Peer reviewed International Research Journal, Published by Global Journals Inc, USA, Vol.12, No. 1, January-2012.