Development of Thermo Plastic Gears for Heavy Duty Applications Using APDL

[1]S.Rajesh Kumar, [2]Dr.M.Kannan and [3]V.Bakkiyaraj,
[1]PG Scholar, [2]Professor, [3]Assistant Professor, [1,2,3]Department of Mechanical Engineering,
[1,3]SKP Engineering College, Tiruvannamalai, [2]M.A.M. College of Engineering, Trichy, India

Abstract: The aim of this project to evaluate the contact stress of plastic material in gear for heavy duty loading condition. Plastic gears have good qualities like self-lubrication, low density, corrosion resistance and smooth running besides its lower strength. Plastic materials provide adequate strength with weight reduction and they are emerging as a better alternative for replacing metallic gears. In this work various plastic materials used. The Plastic materials considered as Poly Ether Ether Ketone (PEEK),Poly Butylenes Terephthalate (PBT), Poly Amide (PA), Poly carbonate (PC) materials is used. These plastic materials have semi crystalline thermo plastic with excellent mechanical and chemical resistance properties that are retained to high temperatures. An APDL gear model has been developed for the design evaluation and comparison study. Gear design parameter like module, face width and pitch circle radius etc can be varied to optimize the design parameter in the geared APDL model to make plastics as feasible gear model. The contact stress of plastic spur gears are calculated by Finite Element Analysis using FEA software ANSYS 13.0.Contact Stress of various plastic gears was analysed in this project.

Keywords: *Plastics Spur gear, ANSYS APDL, Contact Stress, Finite Element Analysis.*

I. INTRODUCTION

The spur gears when in action, are subjected to several stresses but out of all, two types of stresses viz., bending stress and contact stress are important from the design point of view. The bending stresses are theoretically analysed by Lewis equation and contact stresses by Hertz equation. Since spur gears have complicated geometry, a need arises for improved analysis using numerical methods which provide more accurate solutions than the theoretical methods. Finite Element Analysis is one such method which has been extensively used in analysis of components used in various mechanical systems.

Plastic gears have positioned themselves as serious alternatives to traditional metal gears in a wide variety of applications. The use of plastic gears has expanded from low power, precision motion transmission into more demanding power transmission applications. As designers push the limits of acceptable plastic gear applications, more is learned about the behaviour of plastics in gearing and how to take advantage of their unique characteristics.

Plastic gears provide a number of advantages over metal gears. They have less weight, lower inertia and run much quieter than their metal counterparts. Plastic gears often require no lubrication or can be compounded with internal lubricants such as silicone. Plastic gears usually have a lower unit cost than metal gears, and can be designed to incorporate other features needed in the assembly (part consolidation).These gears are also resistant to many corrosive environments.

The plastic materials have corrosion resistance, low electrical and thermal conductivity, easily formed into complex shapes, wide choices of appearance, colours and transparencies. The introduction of plastic materials was made it possible to reduce the weight of the spur gear without any reduction on load carrying capacity and stiffness.

The Finite element analysis of spur gears has been reported by some researchers in the recent past. Bharat Gupta et al., [1] reported the contact stress analysis of spur gears and concluded that the contact stress forms the basis of resisting the gear pitting failure. They also concluded that module is an important parameter for gear design and showed that for the transmission of large power and minimization of contact stress, a spur gear with higher module must be preferred. T. Shoba Rani et al., [2] have performed finite element analysis on spur gear using different materials viz., nylon, cast iron and polycarbonate. They observed that in order to get good efficiency, life and less noise cast iron gears can be replaced by nylon gears because of the fact that the deflection of cast iron is more than that of nylon. Sushil Kumar Tiwari et al., [3] analysed the contact stress and bending stress of involute spur gear teethby FEA and compared the same with Hertz equation, Lewis equation and AGMA/ANSI equations. The FEA results showed a good degree of agreement with the theoretical results.

V. Siva Prasad et al., [4] This paper describes design and analysis of spur gear and it is proposed to substitute the metallic gears of sugarcane juice machine with polymer gears to reduce the weight and noise. A virtual model of spur gear was created in PRO-E, Model is imported in ANSYS 10.0 for analysis by applying normal load condition. The main purpose of this paper to analysis the different polymer gears namely nylon, polycarbonate and their viability checked with counterpart metallic gear like as cast iron. Concluding the study using the FEA methodology, it can be proved that the composite gears, if well designed and analysed, will give the useful properties like as a low cost, noise, Weight, vibration and perform its operation similar to the metallic gears. Based on the static analysis Nylon gear are suitable for the application of sugarcane juice machine under limited load condition in comparison with cast iron spur gears.

Vivek Karaveer et al., [5] This paper presents the stress analysis of mating teeth of the spur gear to find maximum contact stress in the gear tooth. The results obtained from finite element analysis are compared with theoretical Hertz equation values. The spur gear are modelled and assembled in ANSYS DESIGN MODELER and stress analysis of Spur gear tooth is done by the ANSYS 14.5 software. It was found that the results from both Hertz equation and Finite Element Analysis are comparable. From the deformation pattern of steel and grey cast iron, it could be concluded that difference between the maximum values of steel and grey CI gear deformation is very less.

Mahebub Vohra et al., [6] In this paper, Metallic material Cast iron and Non Metallic material Nylon are investigated. The stress analysis of the lathe machine headstock gear box isanalysed by finite element analysis. Analytical bending stress is calculated by two formula Lewis formula and AGMA formula. Analytical results are compared with the finite element method result for validation. Concluding the study, we observed that finite element method software ANSYS have values of stress distribution were in good agreement with the theoretical results. Besides non-metallic material can be used instead of metallic material because non-metallic material provide extra benefits like as less cost, self-lubricating, low noise, low vibration and easy manufacturing.

II. MATERIAL PROPERTIES

The material chosen for the study is Plastics. Table 1 shows the properties of various plastic materials as presented in ANSYS APDL 13.0 engineering data sources.

Table 1: Material Properties of various plastics materials

Material Property	PEEK	Acetal	PA	PET	PC	PBT
Young's Modulus (GPa)	3.76	2.3	2.6	2	2.4	2.6
Poisson's ratio	0.37	0.35	0.39	0.37	0.37	0.37
Density (Kg/m^3)	1300	1410	1130	1300	1200	1290
Coefficient offriction	0.18	0.3	0.2	0.3	0.31	0.21

The plastic materials taken for analysis as Poly Ether Ether Ketone (PEEK), Acetal, Poly amide (PA), Poly ethylene Terephthalate (PET), Poly carbonate (PC) and Poly Butylenes Terephthalate (PBT).

III. PARAMETRIC MODELING OF SPUR GEAR PAIR

The spur gear is sketched and modeled in ANSYS APDL 13.0. The parameters of the gear set are given in table 2.

Table 2:Dimensions of the Spur Gears

S. No.	Description	Symbol	Value	Units
1	Module	m	2.5	mm
2	Pressure Angle	ϕ	20	◦
3	No of teeth	Z	60	
4	Center distance	a	150	mm
5	Face width	b	25	mm
6	Pitch Circle Diameter	dp	150	mm
7	Base Circle Radius	rb	70.47	mm
8	Addendum Circle Radius	ra	77.5	mm
9	Dedendum Circle Radius	rd	71.8	mm
10	Addendum	ha	2.5	mm
11	Dedendum	hd	2.89	mm
12	Fillet Radius	rp	0.98	mm
13	Shaft Radius	rs	16	mm
14	Total angle	Ta	360	◦
15	Bottom clearance	c	0.25	

Parametric modeling allows the design engineer to let the characteristic parameters of a product drive the design of that product. In this paper module, pressure angle, numbers of teeth of spur gear are taken as input parameters. Ansys uses these parameters, in combination with its features to generate the geometry of the Spur gear and all essential information to create the model. By using the relational equation in Ansys APDL, the accurate three dimensional Spur gear model is developed. The parametrical process can increase design accuracy, reduce lead times and improve overall engineering productivity.

Spur Gear Modeling Three Teeth:

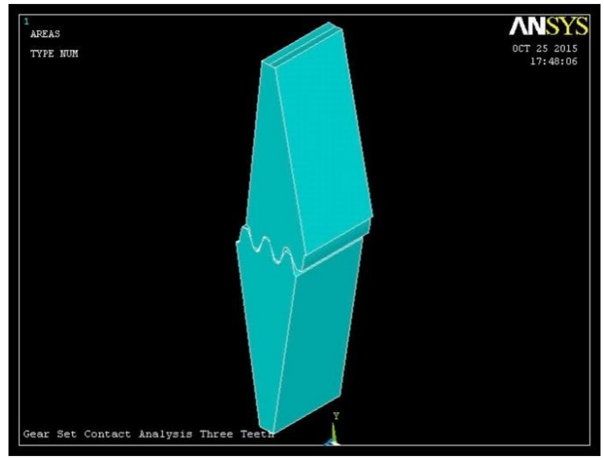


Figure 1: Three Dimensional Model of Spur Gear

IV. SPUR GEAR DESIGN

Specifications of gear:

Module (m) = 2.5 mm
Center distance (a) = 150 mm
Pressure angle (α) = 20 degree
Power (P) = 2.25 Kw
Speed (N) = 750 rpm

Design Calculation:

Power (P) = 2*π*N*T / 60
2250 = (2*π*750*T) / 60
T = (2250*60)/ (2*π*750)
Torque = 28.64788 N-m
Torque T = F * (d/2)
Force F = T/ (d/2)
= 28.64788/0.075
Force F = 381.97185 N

Calculation of tangential load:

$$F_t = \frac{P}{V} * K_0$$

Where, P= power
v= pitch line velocity
K_o = Service factor= 1.25 (medium)

$$V = \frac{\pi * D_1 * N_1}{60}$$

$$V = \frac{\pi * m * 60 * 750}{60 * 1000}$$

v = 2.356m m/s

$$F_t = \frac{2.25 * 1000}{2.356m} * 1.25$$

$$F_t = \frac{1193.761}{m}$$

Calculation of initial dynamic load:

$$F_d = F_t * C_v$$

C_v= Velocity factor

$$C_v = \frac{1 + v_m}{1 + 0.25v_m}$$

$$C_v = \frac{1 + 12}{1 + 0.25 * 12}$$

$$C_v = 3.250$$

$$F_d = \frac{1193.761}{\mathrm{m}} * 3.250$$

$$F_d = \frac{3879.722}{\mathrm{m}}$$

Calculation of beam strength:

$$F_s = \pi m\, b\, \sigma_b\, y$$

Face width b = 10m

Allowable static stress σ_b = 58.86 N/m^2

Form factor = y

$$y = 0.154 - \frac{0.912}{z}$$

y = 0.139

F_s =π *m* 10m* 58.86* 0.139

F_s =257.031m^2

Calculation of module:

$$F_s \geq F_d$$

$$257.031\ \mathrm{m}^2 \geq \frac{3879.722}{\mathrm{m}}$$

m^3 = 15.094

m = 2.471

Take higher standard module m = 2.5

Then,

Face width b = 10m = 25mm

Pitch circle diameter d_1= m *z_1= 150mm

Pitch line velocity v = 5.890 m/s

Recalculation of beam strength:

$$F_s = \pi m\, b\, \sigma_b\, y$$

$$F_s = \pi * 2.5 * 25 * 58.86 * 0.139$$

$$F_s = 1606.44\ \mathrm{N}$$

Calculation of accurate dynamic load:

$$F_d = F_t * C_v$$

$$F_t = \frac{P}{V} = \frac{2.25 * 1000}{5.890}$$

$$F_t = 381.97\ \mathrm{N}$$

$$C_v = \frac{1 + v}{1 + 0.25v}$$

$$C_v = \frac{1 + 5.890}{1 + 0.25 * 5.890}$$

$$C_v = 2.787$$

$$F_d = 381.97 * 2.787$$

$$F_d = 1064.51\ \mathrm{N}$$

Check for beam strength (tooth breakage):

$$F_s > F_d$$

1606.44> 1064.51

The design is safe.

Calculation of wear load:

$$F_w = d_1 * b * Q * k_w$$

Q = Ratio factor

$$Q = \frac{2i}{i + 1}$$

$$Q = \frac{2 * 1}{1 + 1}$$

Q = 1

k_w =Load stress factor =1.4 N/m^2(For non-metallic gear)

$$F_w = 150*25*1*1.4$$

$$F_w = 5250\ \mathrm{N}$$

Check for wear:

$$F_w > F_d$$

5250> 1064.51

The design is Safe. Safe for surface durability.

V. FINITE ELEMENT ANALYSIS USING ANSYS APDL 13.0

Finite element modeling is described as the representation of the geometric model in terms of a finite number of element and nodes. It is actually a numerical method employed for the solution of structures or a complex region defining a continuum. Solutions obtained by this method are rarely exact for analysis in ANSYS 13.0. The loading conditions are assumed to be static. Load applied at rotational pressure inside driving gear rim and DOF applied at driven pinion inside rim.

To simulate gear contact surfaces, contact conditions are a special class of discontinuous constraint, allowing forces to be transmitted from one part of the model to another. The constraint is discontinuous because it is applied only when the two surfaces are in contact. When the two surfaces separate, no constraint is applied. The analysis has to be able to detect when two surfaces are in contact and apply the contact constraints accordingly. An accurate contact simulation can be achieved using the latest non-linear FEA techniques by defining the contact surfaces of the teeth as the contact pairs which are special elements designed to treat cases of changing mechanical contact between the parts of an assembly or between different faces of a single part in ANSYS software.

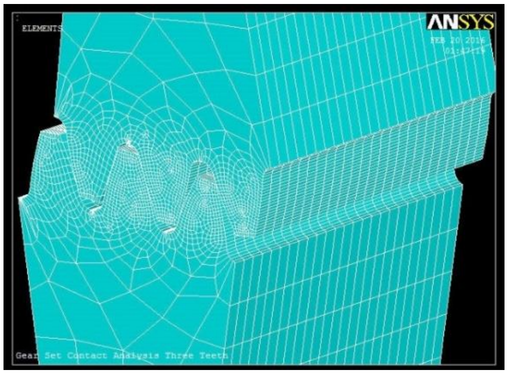


Figure 2: Meshing of spur gear

To apply finite element method in contact stress a special technique was used rather the regular elements, to distinguish between the contact regions which were in two parts. One was the first body named target region and the other body was named contact region. For target region, target elements were used and in contact region contact elements were used. ANSYS software presents a significant technique for this purpose which was used here.

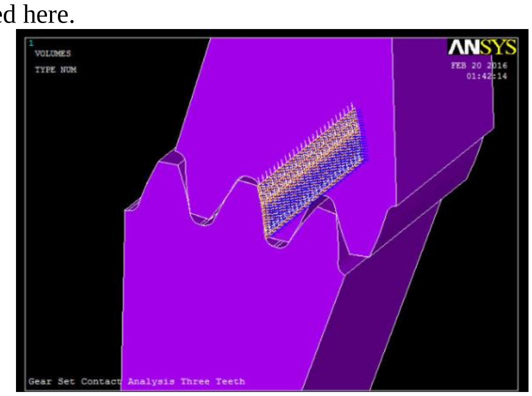


Figure 3: Contact pair of spur gear

A computer program was developed to plot one pair of teeth in contact. This Program used to build finite element contact models and contact finite element analysis was done under the load and material conditions were named. In gear meshing, the contact of the teeth is a surface-surface contact and the profiles of the teeth are curves. Therefore, TARGE170 and CONTA174 are chosen as the contact elements to represent the contact pairs. TARGE170 is used to represent the 3-D target surfaces for the associated contact elements. This target surface is discretized by a set of target segment elements. CONTA174 is used to represent contact and sliding between 3-D "target" surfaces. Contact occurs when the element surface penetrates one of the target segment elements on a specified target surface.

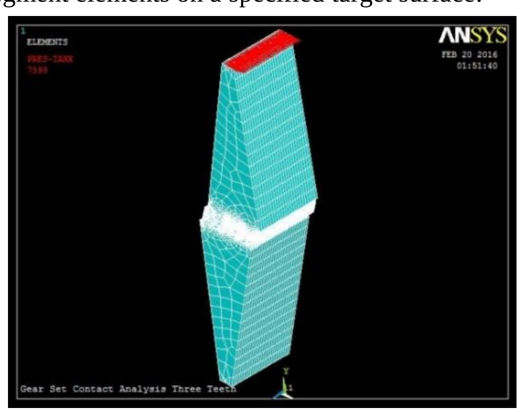


Figure 4: Boundary conditions of spur gear

In the present study, FEA software ANSYS 13.0 APDL has been used to determine the maximum allowable contact stress in plastics spur gears. Fine meshing as shown in figure 2, has been done in order to get accurate results. A moment of 7599 N-m is applied in clockwise direction on the inner rim of the upper gear.

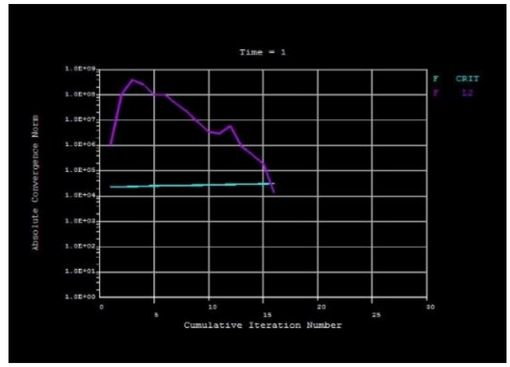


Figure 5: Non-linear convergence graph

VI. RESULTS AND DISCUSSIONS

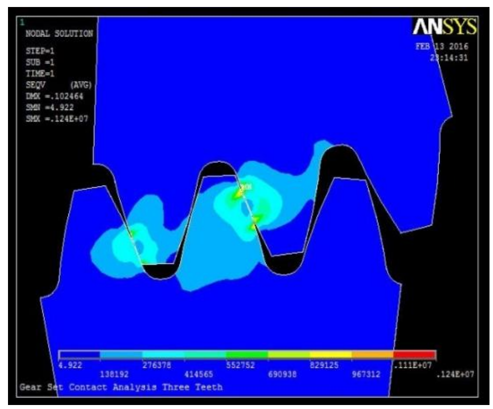


Figure 6: Stress distribution

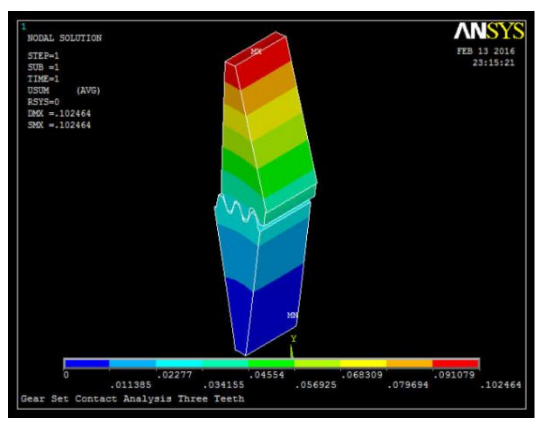


Figure 7: Deformation pattern

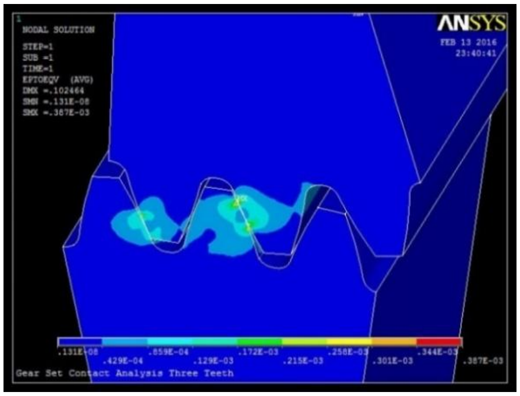


Figure 8: Strain

As is clearly indicated in figure 6, the maximum allowable contact stress for plastic spur gears of dimensions given in table 2 and for transmitted torque of 28.94 N-m is 1.24MPa, as determined by FEA using ANSYS APDL 13.0.

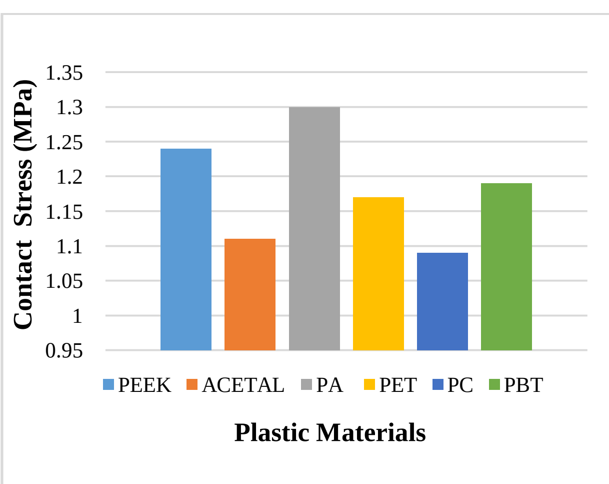
Analysis Result for Various Plastic Gear Materials:

Table 3: Analysis results tabulation

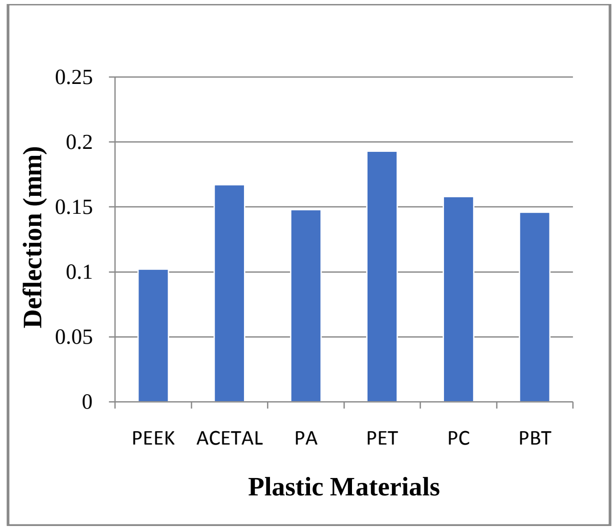
S.No	Plastic Materials	Contact Stress (MPa)	Deflection (mm)
1	Peek	1.24	0.102
2	Acetal	1.11	0.167
3	PA	1.30	0.148
4	PET	1.17	0.193
5	PC	1.09	0.158
6	PBT	1.19	0.146

From the static analysis using ansys the deflections and vonmise stress values for the Poly Ether Ether Ketone (PEEK), Acetal, Poly amide (PA), Poly ethylene Terephthalate (PET), Poly carbonate (PC) and Poly Butylenes Terephthalate (PBT) and polycarbonate materials are obtained as following above tables.

Graph 1: Contact stress Vs various plastic materials



Graph 2: Deflection Vs various plastic materials



CONCLUSION

In this project we have to calculate the contact stress of plastic gear materials. The finite element modeling and analysis of spur gear is done to determine the contact stress by ANSYS 13.0 APDL.

In the present study, we report the contact analysis of 20 degree full depth involute plastic spur gears during the transmission of power of 2.25kW by FEA using ANSYS. A general mathematical model is proposed for evaluating the contact stress in plastic spur gears of equal geometry in mesh using FEA. Maximum contact stress and deflection for various plastic gear materials are obtained in this study.

From the deformation pattern of various plastic materials, it could be concluded that Peek gear deformation is very less. According to the study, analysis, results and Graphs it was concluded that the plastic gear can be used for heavy duty applications.

References

[1] Gupta, Mr Bharat, Mr Abhishek Choubey, and Mr Gautam V. Varde. "Contact stress analysis of spur gear." In International Journal of Engineering Research and Technology, vol. 1, no. 4 (June-2012). ESRSA Publications, 2012.

[2] Rani, T.Shoba, and T.Dada Khalandar. "SPUR GEAR." International Journal of Computational Engineering Research 3, no. 11 (2013): 7-12.

[3] Tiwari, Sushil Kumar, and Upendra Kumar Joshi. "Stress analysis of mating involute spur gear teeth." In International Journal of Engineering Research and Technology, vol. 1, no. 9 (November-2012). ESRSA Publications, 2012.

[4] Karaveer, Vivek, Ashish Mogrekar, and T. Preman Reynold Joseph. "Modeling and Finite Element Analysis of Spur Gear." International Journal of Current Engineering and Technology ISSN (2013): 2277-4106.

[5] Lee, Huei-Huang. Finite element simulations with ANSYS workbench 14. SDC publications, 2012.

[6] Raja Roy, M., S. Phani Kumar, and D.S. Sai Ravi Kiran. "Contact Pressure Analysis of Spur Gear Using FEA." International Journal of Advanced Engineering Applications 7, no. 3 (2014): 27-41.

[7] Bhandari, V. B. Design of machine elements. Tata McGraw-Hill Education, 2010.
[8] Gopinath, K., and M. M. Mayuram. "Spur Gear Design." Nptel. Accessed February 08, 2015. http://nptel.ac.in/courses/IIT-MADRAS/Machine_Design_II/pdf/2_8.pdf.
[9] "Spur Gears." Khkgears. Accessed February 08, 2015. www.khkgears.co.jp/world/ break/ Spur tech.pdf.
[10] "Finite Element Method." Wikipedia. Accessed February 14, 2015. http://en.wikipedia.org/wiki/Finite_element_method.
[11] Vivek KaraveerȦ*, Ashish MogrekarȦ and T. Preman Reynold JosephȦ, " Modelling and Finite Element Analysis of Spur Gear", Dec 2013, International Journal of Current Engineering and Technology, Vol 3.
[12] Mahebub Vohra, Prof. Kevin Vyas, "Comparative Finite Element Analysis of Metallic and non Metallic spur gear", May-June 2014, IOSR Journal of Mechanical and Civil Engineering, Vol-11/3,pp- 136-145.
[13] V. Siva Prasad, Syed Altaf Hussain, V. Pandurangadu, K. PalaniKumar, " Modeling and Analysis of spur gear for Sugarcane Juice Machine under Static Load Condition by Using FEA",July-Aug 2012,International Journal of Modern Engineering Research,Vol- 2/4, pp- 2862-2866.