Analysis of differential crown gear and pinion for axle with different materials

[1]Dr. Satypal T. Warghat, [2]Dhiraj V. Astonkar, [3]Manish P. Bijwe

[1]Head of Department, [2,3]Assistant Professor
Department of Mechanical Engineering,
Dr. Sau. KamaltaiGawai Institute of Engineering and Technology,
Darapur, Tq. Daryapur, Dist. Amravati, Maharashtra 444814

***Abstract*: The differential can be stated as a gear train used to control the speed and torque to the rear wheels. While taking a turn, the basic requirement of the vehicle is to control the speed of rear wheels so that the vehicle turns smoothly on road surface.**

The differential mainly consists of three shafts and gear train arrangement. The first shaft is propeller shaft which provides the necessity torque and speed to differential for turn. The other two shafts are axle shafts mounted for each rear wheels. These shafts are attached to propeller shaft by crown gear and pinion arrangement thus making a bevel pair of gears.

The two main parts of differential transmission system, one is crown gear which gives allowable speed to turn the vehicle and another is pinion which provides allowable speed and torque for turning the vehicle. Thus, the analysis of such crown gear and pinion makes necessity for strength.

The analysis is conducted to verify the best material for the gears in the gear box at high speeds by analyzing von miss stress, deformation and also by considering safety of transmission. The various materials selected for analysis are Grey Cast Iron, Structural Steel, Titanium Alloy and Polyethylene. The crown gear and pinion are made in solid modeling software CATIA and analyzed in software ANSYS 15.0 for obtaining the Von Miss Stresses in gear and pinion and Total Deformation in the assembly. The comparison has made based on the results of finite element analysis.

The study is useful in determining the best material for Crown Gear and Pinion. It is expected that the Grey Cast Iron material is most useful for power transmission because it holds moderate properties as compare with Structural Steel, Titanium Alloy and Polyethylene.

***Keywords*: Crown Gear, Crown Pinion, Analysis, FEA, ANSYS, Material etc.**

I. INTRODUCTION

A. Concept

Differential is used when a vehicle takes a turn, the outer wheel on a longer radius than the inner wheel. The outer wheel turns faster than the inner wheel that is when there is a relative movement between the two rear wheels. If the two rear wheels are rigidly fixed to a rear axle the inner wheel will slip which cause rapid tire wear, steering difficulties and poor load holding.

Differential is a part of inner axle housing assembly, which includes the differential rear axles, wheels and bearings. The differential consists of a system of gears arranged in such a way that connects the propeller shaft with the rear axles. The analysis is conducted to verify the best material for the gears in the gear box at higher speeds by analyzing stress, displacement and also by considering safety of transmission.

The bevel gears are used for transmitting power at a constant velocity ratio between two shafts whose axes intersect at a certain angle. The pitch surfaces for the bevel gear are frustums of cones. The two pairs of cones in contact are shown in Fig. 1.1. The elements of the cones, as shown in Fig. 1.1 (a), intersect at the point of intersection of the axis of rotation. Since the radii of both the gears are proportional to their distances from the apex, therefore the cones may roll together without sliding. In Fig. 1.1 (b), the elements of both cones do not intersect at the point of shaft intersection. Consequently, there may be pure rolling at only one point of contact and there must be tangential sliding at all other points of contact. Therefore, these cones, cannot be used as pitch surfaces because it is impossible to have positive driving and sliding in the same direction at the same time. Thus, it concludes that the elements of bevel gear pitch cones and shaft axes must intersect at the same point.

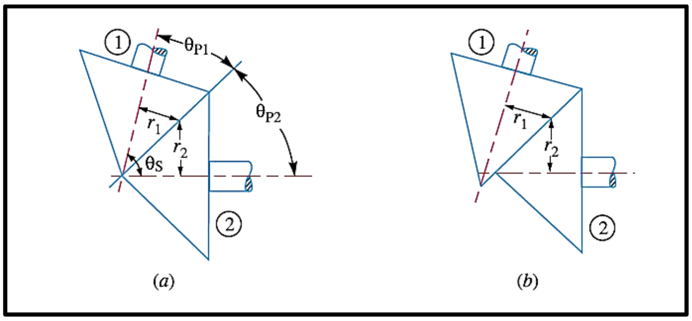

Figure 1.1 Pitch surface for bevel gears

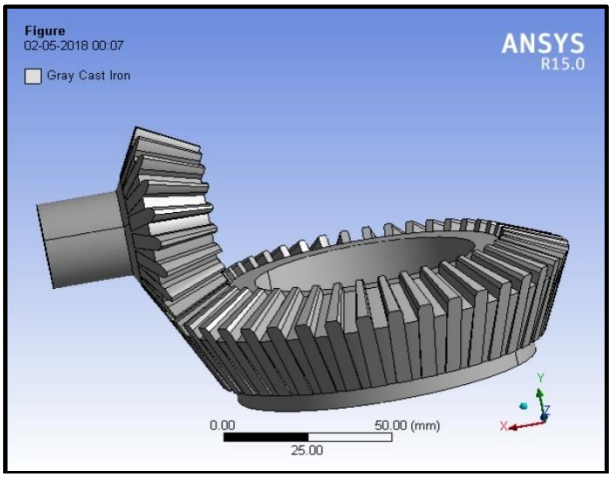

Figure 1.2 A Pair of Crown Gear and Pinion

B. Objectives of Work

Following are the various objectives numerated from analysis of a crown gear and pinion.

1. To understand the power transmission in automotive vehicles through differential.
2. To design the crown gear and pinion
3. To obtain the optimized material selection for the design and analysis.
4. To compare stresses generated by torque and speed of vehicles.

II. FINITE ELEMENT METHOD

A. Introduction to FEA

Traditional approach to design analysis involves the application of classical or analytical techniques. This approach has the following limitations:

i. Stresses and strains are obtained only at macro level. This may result in inappropriate deployment of materials. Micro level information is necessary to optimally allocate material to heavily stressed parts.
ii. Adequate information will not be available on critically stressed parts of the components.
iii. It may be necessary to make several simplifications and assumptions to design complex components and systems, if design analysis is carried out in the conventional manner.
iv. Manual design is time consuming and prone to errors.
v. Design optimization is tedious and time consuming.

FEA is a convenient tool to analyze simple as well as complex structures. The use of finite element analysis is not restricted to mechanical engineering systems alone. FEA finds extensive application in electrical engineering, electronics engineering, micro electromechanical systems, biomedical engineering etc. In manufacturing, FEA is used in simulation and optimization of manufacturing processes like casting, machining, plastic molding, forging, metal forming, heat treatment, welding etc. Structural, dynamic, thermal, magnetic potential and fluid flow problems can be handled with ease and accuracy using FEA.
FEA was initially developed in 1943 by R. Courant to obtain approximate solution to vibration problems. Turner et al published in 1956 a paper on "Stiffness and Deflection of Complex Structures". This paper established a broader definition of numerical analysis as a basis of FEA. Initially, finite element analysis programs were mainly written for mainframe and mini computers. With the advent of powerful PC's, the finite element analysis could be carried out with the help of several FEA software packages. Finite element method can be applied to a variety of design problems concerning automobiles, airplanes, missiles, ships, railway coaches and countless other engineering and consumer products.
The finite element method is a numerical method, which can be used for the solution of complex engineering problems with accuracy acceptable to engineers. In 1957 this method was first developed basically for the analysis of aircraft structures. There after the usefulness of this method for various engineering problems were recognized. Over the years, the finite element technique has been so well developed that, today it is considered to be one of the best method for solving a wide variety of practical problems efficiently. One of the main reasons for the popularity of the method in different fields of engineering is that once a general computer program is written, it can be used for the solution of any problem simply by changing the input data.
In FEM since the actual problem is replaced by a simpler one in finding the solution we will be able to find only an approximate solution rather than the exact solution. In most of the practical problems, the existing mathematical tools are not even able to find approximate solution of the problem. Thus, in the absence of any other convenient method to find even the approximate solution of a given problem, we have to prefer the FEM. The digital computer provided a rapid means of performing many calculations involved in FEA. Along with the development of high speed computers, the application of the FEM also progressed at a very impressive rate.

B. Steps for Finite Element Analysis

The basic steps adopted for analyzing an engineering problem by the FEM are,

i) Discretization of the Continuum: - The continuum is the body, structure or solid being analyzed. Discretization is the process of dividing the body into a finite number of elements, as shown in Figure. These elements may have different sizes. The choice of the element type, its shape and refinement are required to be decided before discretization. The success of the discretization, which is also known as meshing, lies in how closely the discredited

Figure 1.3 Non Discretized Body

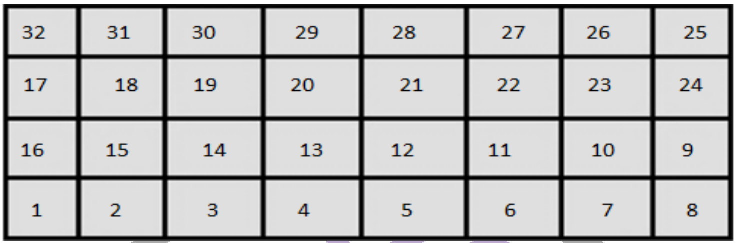

Figure 1.4 Discretized Body

Continuum represents the actual continuum. The size and orientation of the elements are selected in such a manner that an element is exclusively composed of only one material. Such type of continuum for beam model is shown in Figure 5.1 and 5.2.

ii) Formation of Element Stiffness Matrix: -After the continuum is discretized with the desired type of element, the element stiffness matrixes are formulated for all the elements. The elements stiffness matrix, which depends upon the material and geometry of an element, can be formulated by following methods,

a) Direct Stiffness Methods and

b) Energy Method

In some problems, such as trusses, the local coordinate system of an individual element is different from the global coordinate system. In such cases, the element stiffness matrix in local coordinate system $[k']_e$ is converted to the element stiffness matrix in global coordinate system $[k]_e$ by using the transformation matrix [L]. The element stiffness matrix in global coordinate system is given by,

$[k]_e = [L]^T [k']_e[L]$

iii) Formation of Global Stiffness Matrix: -After element stiffness matrix in global coordinate system is obtained for all the elements, they are assembled to form the global stiffness matrix [K] for the entire body.

$[K] = \Sigma [k]_e$

The global stiffness matrix is symmetric, banded and singular.

iv) Formation of Global Load Vector: -The element force vectors in the global coordinate system $\{f\}_e$ for all elements are assembled to form the global vector {F} for the entire body.

$\{F\} = \Sigma \{f\}_e$

v) Formation of Global Node Displacement Vector: -The global nodal displacement vector $\{U_N\}$ is formed for the entire body.

vi) Assembly of Global stiffness vector -The relation between the global stiffness matrix [K], global nodal displacement vector $\{U_N\}$ and the global load vector {F} is expressed as a set of simultaneous algebraic equation.

$[K] \{U_N\} = \{F\}$

vii) Incorporation of Specified Boundary conditions: -The specified boundary conditions are incorporated in equilibrium equation by using one of the following two approaches:

a) The Elimination approach and

b) The Penalty approach

viii) Solution of Simultaneous Equations:-After including the specified boundary conditions in equilibrium equation, the modified equation are solved for the unknown nodal displacements by using methods like Gaussian elimination, Cholsky's Factorization, Gauss Seidel, Jacobi iterations, Frontal Technique, etc. In solving the equations, advantage is taken of the banded property of the global stiffness matrix.

ix) Computation of element strains and stresses: -Knowing the nodal displacements, the element strains and stresses are calculated. The components of strains at any point within the element are given by,

$\{\varepsilon\} = [B]\{U_N\}$

Similarly, the components of stresses at any point within the element are given by

$\{\sigma\} = [D]\{\varepsilon\}$

Where, [B] = Element strain-nodal displacement matrix

[D] = Element stress-strain matrix.

C. FEA SOFTWARE PACKAGES AND APPLICATIONS

The rapid advance made in computer hardware and software led to significant developments in FEA software. FE programming gas emerged as a specialized discipline which requires knowledge and experience in the diverse areas such as FE technology including foundations of machines, and numerical analysis on the one hand and the computational skills in areas of software technology including programming techniques, data structure, data base management and computer graphics on the other hand. It requires several man years to develop general purpose finite element analysis software with a processing capability and facility for the user to have a wide choice of several types of elements, analysis for different types of problem- static, dynamic, material and geometric nonlinear, coupled situations, hear transfer, interaction problems etc. and pre and post-processing features.

In 1975 the FEA software package was used for the analysis of aero plane parts. From the analysis it was predicted that the ansys packages gives better solution as compare to defining theory models. Thus the packages cover the industrial sector for analysis of various structural and mechanical parts. The main purpose of such packages is to obtain the solution without creating physical model. The figure 5.3 shows memory allocation by ANSYS software package.

The analysis software packages are applicable to static linear analysis, non-linear analysis, modal analysis, fluid flow analysis, thermal analysis, harmonic response analysis, steady state analysis, etc.

As it is not possible here to review the capabilities and compare different commercially available finite element analysis packages only the names of some of the popular packages are given below:

ABAQUS, ADINA, ANSYS, ALGOR, ASKA, COSMOS, GT-STRUDL, LISA, NISA, ASTRAN, PAFEC, PASTRAN, SAP.

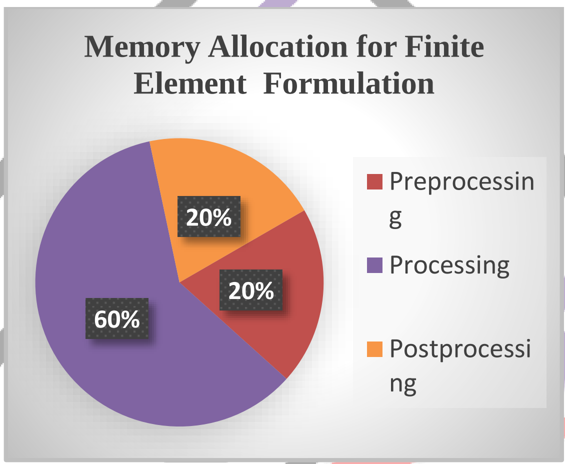

Figure 1.5 Memory Allocations for Analysis Software Package

III. Structural Analysis of Crown Gear and Pinion

Finite element analysis has been carried out by ANSYS15 software. ANSYS is a general-purpose finite-element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis (both linear and nonlinear), heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.

Here the assembly is discretized into approximately 8671 elements having 15700 nodes by default meshing of tetrahedral and mesh size is 2 mm. The solution is obtained by using ANSYS Work Bench modeler. The maximum stress is found out and from the resulted data, the comparison has made among the all types of torque.

Following steps show the guidelines for carrying out Structural Analysis.

Define Materials

1. Set preferences. (Structural steel)
2. Define constant material properties. (Density, Young's modulus, Poisson's ratio)

Model the Geometry

3. Follow bottom up modeling and create/import the geometry.

Generate Mesh

4. Define element type. (Default mesh of element size 2 mm)
5. Mesh the area.

Apply Boundary Conditions

6. Apply constraints to the model.

Obtain Solution

7. Specify analysis types and options. (Structural ANSYS for maximum Stress and total deformation)
8. Solve

The ANSYS 15 finite element program was used for Structural analysis of crown and pinion. For this purpose, the total 4 models are created depending on torque in CAD software (CATIA) and imported in ANSYS (.stp file). Each model have torque and loading. The materials used for analysis has following properties:

Table 5.1 Various Materials Properties for FEM

Sr. No.	Property	Grey Cast Iron	Structural Steel	Titanium Alloy	Polyethylene
1	Density (Kg/m^3)	7200	7850	4620	950
2	Ultimate Strength (MPa)	240	250	1070	33
3	Young's Modulus (MPa)	1.1e05	200000	96000	1100
4	Poisson Ratio	0.28	0.30	0.36	0.42
5	Bulk Modulus (MPa)	83333	16667	114290	2291.7

A. FEA Results for Grey Cast Iron

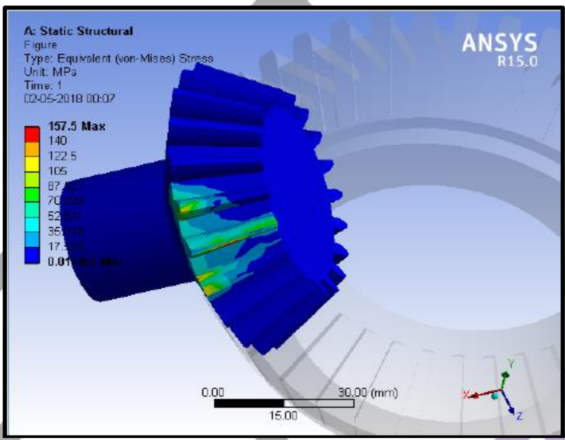

Figure 1.6 Stress of Grey Cast Iron Pinion

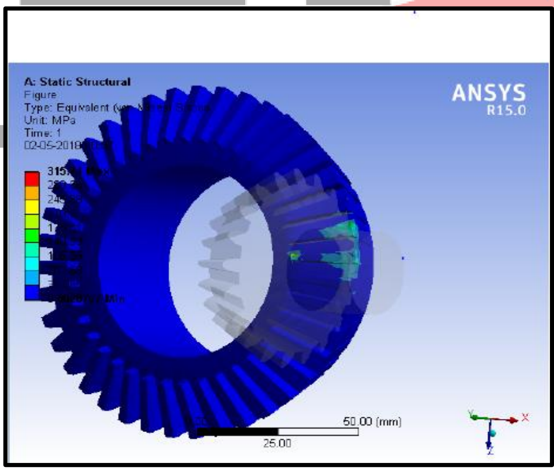

Figure 1.7 Stress of Grey Cast Iron Crown Gear

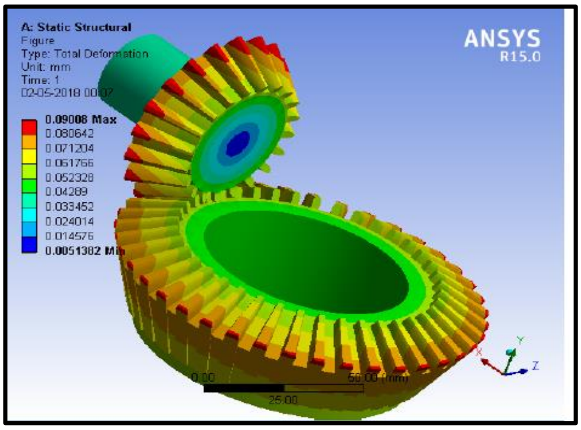

Figure 1.8 Total Deformation of Grey Cast Iron Assembly

B. FEA Results of Structural Steel

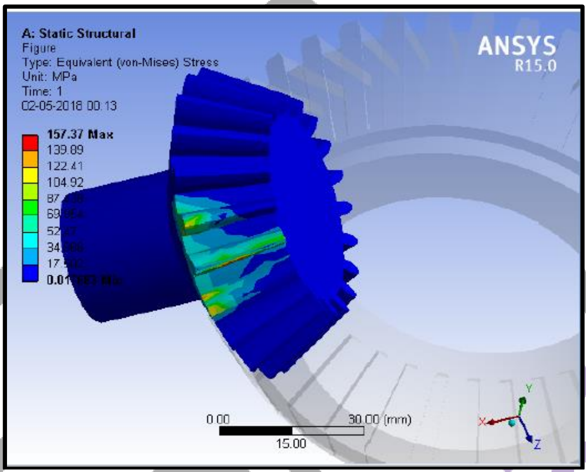

Figure 1.9 Stress of Structural Steel Pinion

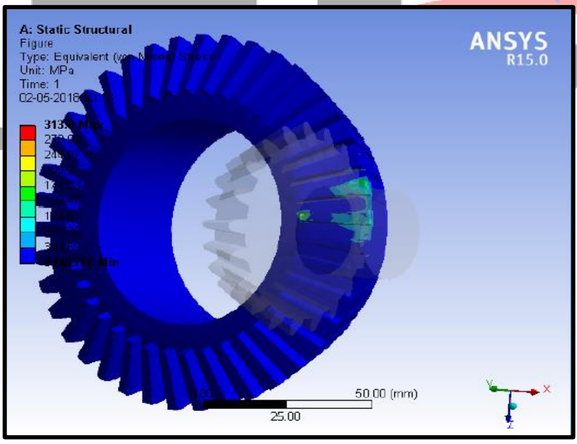

Figure 1.10 Stress of Structural Steel Crown Gear

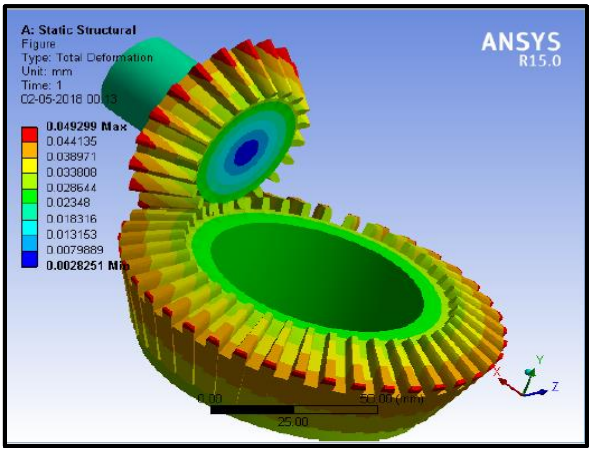

Figure 1.11 Total Deformation of Structural Steel Assembly

C. FEA Results of Titanium Alloy

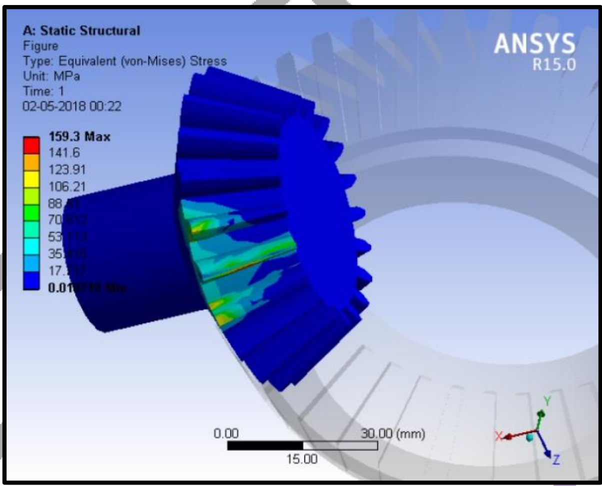

Figure 1.12 Stress of Titanium Alloy Pinion

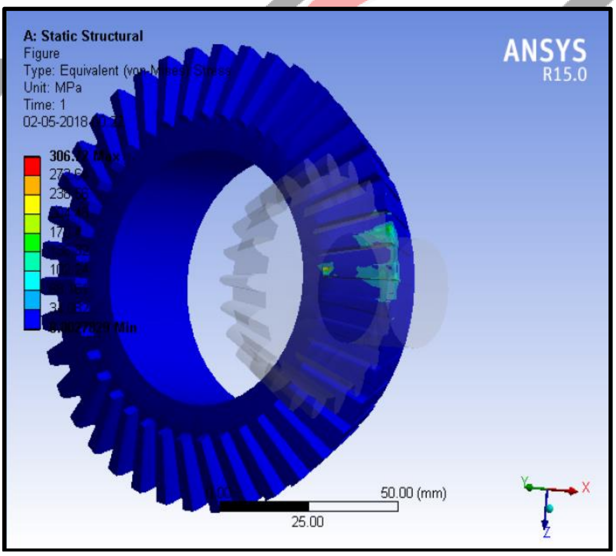

Figure 1.13 Stress of Titanium Alloy Crown Gear

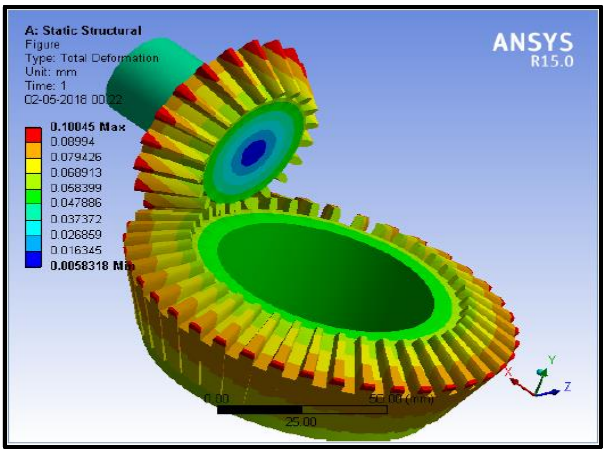

Figure 1.14 Total Deformation of Titanium Alloy Assembly

D. FEA Results of Polyethylene

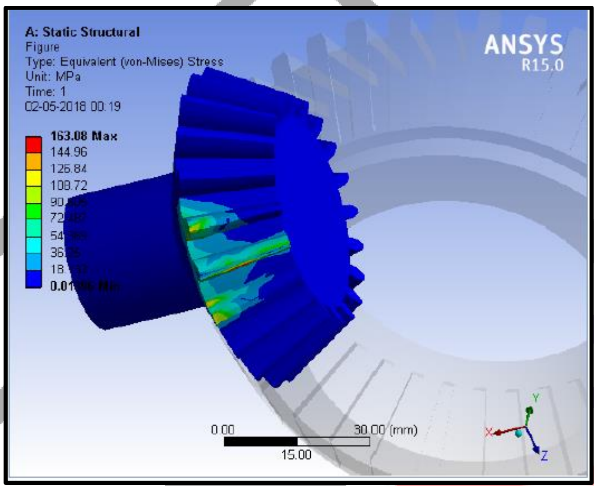

Figure 1.15 Stress of Polyethylene Pinion

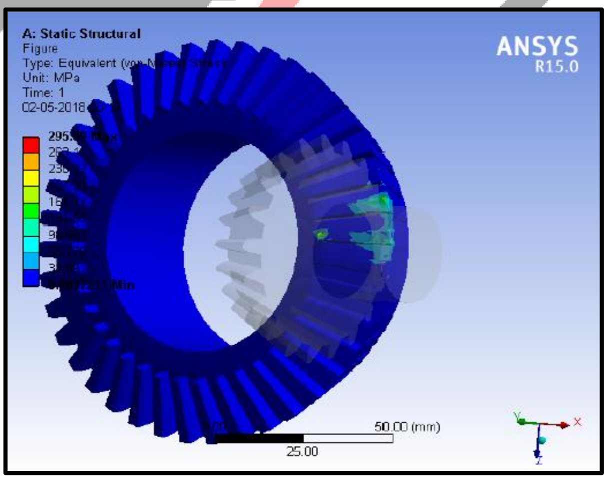

Figure 1.16 Stress of Polyethylene Crown Gear

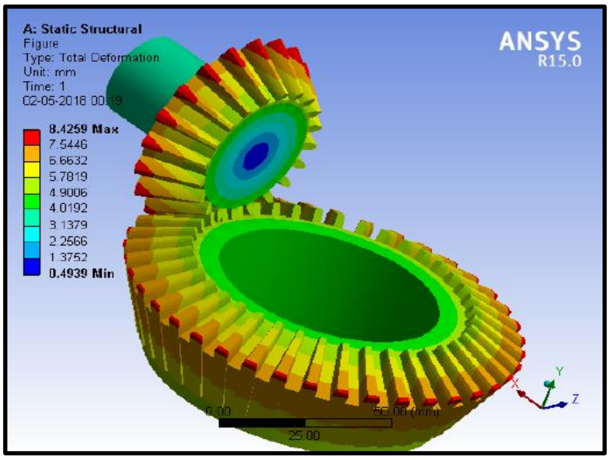

Figure 1.17 Total Deformation of Polyethylene Assembly

IV. Comparison of Results of FEA

Table 5.2 Comparison of Results of FEA

Sr. No.	Material	Part	Stress (MPa)	Total Deformation (mm)
1	Grey Cast Iron	Pinion	157.5	0.092
		Crown Gear	315.34	
2	Structural Steel	Pinion	157.37	0.049
		Crown Gear	313.24	
3	Titanium Alloy	Pinion	159.3	0.100
		Crown Gear	306.22	
4	Polyethylene	Pinion	163.08	8.425
		Crown Gear	295.59	

V. CONCLUSION

The present study includes the design and analysis of crown gear and pinion assembly by selecting various materials and draws following concluding remarks:

1. The power transmission in automotive vehicles through differential have been understood successfully.
2. The assembly of crown gear and pinion have been designed in CATIA and analyzed in ANSYS 15.0 Software by selecting various materials.
3. From the Finite Element Analysis it is concluded that the Grey Cast Iron assembly of crown gear and pinion is most useful as compare with other material assemblies. Since, the maximum stress in pinion is 157.5 MPa and in gear is 315.34 MPa. Also the total deformation in assembly is 0.09 mm.
4. Thus, it is good agreement that, the grey cast iron material is most helpful as compare to Structural Steel, Titanium Alloy and Polyethylene when Crown Gear and Pinion Assembly is considered.

REFERENCES

Alcacer, V. (2015). Design, Manufacturing & Analysis of An Abs Differential Gear System Model. *Proceedings of the 6th International Conference on Mechanics and Materials in Design*, 589-599.

Bagewadi, S. S. (2014). Design And Analysis Of Crown Pinion Of A Differential Gear Box For Reduced Number Of Teeth To Improve Torque Transmitted. *International journal of mechanical engineering and robotic research, 3*, 188-197.

Bagewadi, S. S. (2014). Design And Analysis Of Crown Pinion Of A Differential Gear Box For Reduced Number Of Teeth To Improve Torque Transmitted. *international journal of mechanical engineering and robotics research*, 188-199.

Das.A, D. (2013). Structural Analysis of Differential Gearbox. *international journal of innovations in engineering and technology, 2*(1), 65-71.

Joseph Gerald. M, K. C. (2015). Design and Fabrication of Spur Gear. *International Journal of Innovative Research in Science, Engineering and Technology, 4*(2), 22-28.

Kapoor, N. (2016). Design And Stress Strain Analysis Of Composite Differential Gear Box. *International Journal of Science, Engineering and Technology Research*, 188-198.

N.Vijayababu, C. (2015). Design And Analysis Of Differential Gear Box Used In Heavy Vehicle. *Proceedings of International Conference on Recent Trends in Mechanical Engineering, 2*(1), 139-145.

S. H. Gawande, S. K. (2006). Design, Manufacturing & Analysis of Differential Crown Gear. *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, 59-66.

Shashank Pandey, N. N. (2017). Modeling, Design & Analysis Of Differential Gear Box And Its Housing Through Fem, Solidwork & Ansys Benchwork 14.0. *International Journal Of Engineering Sciences & Research*, 127-140.

Sunny, S. (2014). Design Of An Automotive Differential With Reduction Ratio Greater Than 6. *International Journal of Research in Engineering and Technology, 3*(7), 72-83.

T.Loknadh. (2015). Design a crown wheel gear of differential gear box used in heavy vehicles for structural loads. *International Journal of Science, Engineering and Technology Research, 4*(7), 2705-2714.

Teja, N. S. (2017). Design And Analysis Of Differential Gear Box In Automobiles. *International Journal of Mechanical Engineering and Technology (IJMET), 8*(5), 175-186.