Material selection for Herringbone Gear through Modeling and Finite Element Analysis

[1] D.Ramakrishna, [2] S.Arul

[1,2] Department of Mechanical Engineering,

Annamacharya Institute of Technology & Sciences, Tirupati-517520. Andhra Pradesh, India.

Abstract : Herringbone gears are having two sets of helical teeth, one set inclined at an acute angle to the other and are used to transfer large loads without thrust load on the shafts. Owing to the higher load carrying capacity, larger total contact ratio and lower axial force, the herringbone gears are preferred over spur and helical gears in planetary gear trains. They are preferably suited for high shock and vibration applications. Modal analysis and Fatigue analysis have been carried out on herringbone gears of alloy steel and carbon fiber reinforced composite using ANSYS. The results show that the superior material for herringbone gear can be attained by comparing the material properties. This is substantiated by monitoring deformation and fatigue life.

IndexTerms - **Herringbone gear, FEM, Modal analysis and Fatigue analysis.**

1. INTRODUCTION

Gears are mechanical components that transmit rotation and power from one shaft to another in industrial machinery, automobiles, aircrafts, marine vessels etc. Owing to power losses amount to heat generation within the gearbox and gear failure modes such as scoring and fatigue can be directly influenced by the efficiency of the gearing system. To ensure the system operates reliably, the efficiency of these power transmission systems is an important design factor.

Herringbone gears are designed to transmit power through parallel and perpendicular axes. The unique tooth structure of this gear consists of two adjoining, opposite helices that appear in the shape of the letter 'V'. These gears are preferred over spur and helical gears due to higher load carrying capacity, larger total contact ratio, lower axial force, reliable transmission and smooth operation. Therefore, the herringbone gear train is useful for heavy machineries.

Many researchers have carried out the dynamic modeling and dynamic analysis of spur or single-helical gears. J. Lin and Parker (1-2) set up a translation-torsional dynamic model for a single-stage spur planetary gear to study the free vibration. Kahraman (3) developed a single-stage helical PGT dynamic model to investigate the dynamic behavior of a four-planet PGT system. However, the published literature on design and analysis of herringbone gears are limited.

Sondkar et al. (4) created a linear, time-invariant dynamic model of a single-stage double–helical planetary gear set to investigate the free and forced vibration characteristics. Bu et al. (5) developed a dynamic model for HPGT with journal bearings to only explore its modal properties, but they regard herringbone gears as the absence of axial force.

Bending stress, Permissible bending stress, contact stress, bending fatigue strength, allowable surface fatigue stress, tooth surface strength of gear and pinion etc. are to be carefully considered while designing them. To enhance the effectiveness of the gear design, all these metrics are controlled properly. In this paper, solid works is used to model and assemble herringbone gear. Modal analysis and Fatigue analysis have been carried out on herringbone gears of alloy steel and carbon fiber reinforced composite through ANSYS to verify the gear design rationality.

2.0 DESIGNING OF GEAR

- Power, P= 12.3KW (16.5 HP),
- Torque, T = 14.8 N-m,
- Speed, N = 8000 rpm
- Diameter of gear, Dg = 100 mm
- Dg = m x Tg
- Tg = 100/2 = 50 teeth,
- Face width, b = 4 π m = 4x3.14x2 = 25.133 mm
- Tangential force, Wt = 2000 T / Dg =   2000 x 14.8 /100 =   493.33 N
- Addendum = 0.943 x m = 0.943 x 2 = 1.886 mm
- Dedendum = 1.257 x m = 1.257 x 2 = 2.514 mm
- Tooth Height = 2.2 x m = 2.2 x 2 = 4.4 mm
- Tooth Thickness = 1.493 x m = 1.493 x 2 = 2.986 mm
- Fillet Radius   = 0.4 x m = 0.4 x 2 = 0.8 mm

By considering all these dimensions, the double helical gear is designed by using CATIA modeler.

About CATIA

CATIA is a 3D mechanical CAD (computer-aided design) program that runs on Microsoft Windows and is being developed by Dassault Systems Solid works Corp., a subsidiary of Dassault Systems, S. A. Vélizy, France). Parameters refer to constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric parameters, such as line lengths or circle diameters, or geometric parameters, such as tangent, parallel, concentric, horizontal or vertical, etc. Numeric parameters can be associated with each other through the use of relations, which allow them to capture design intent. Operation-based features are not sketch-based, and include features such as fillets, chamfers, shells, applying draft to the faces of a part, etc.

Building a model in Solid works usually starts with a 2D sketch (although 3D sketches are available for power users). Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity and concentricity. The parametric nature of Solid works means that the dimensions and relations drive the geometry, not the other way around. The dimensions in the sketch can be controlled independently, or by relationships to other parameters inside or outside of the sketch.
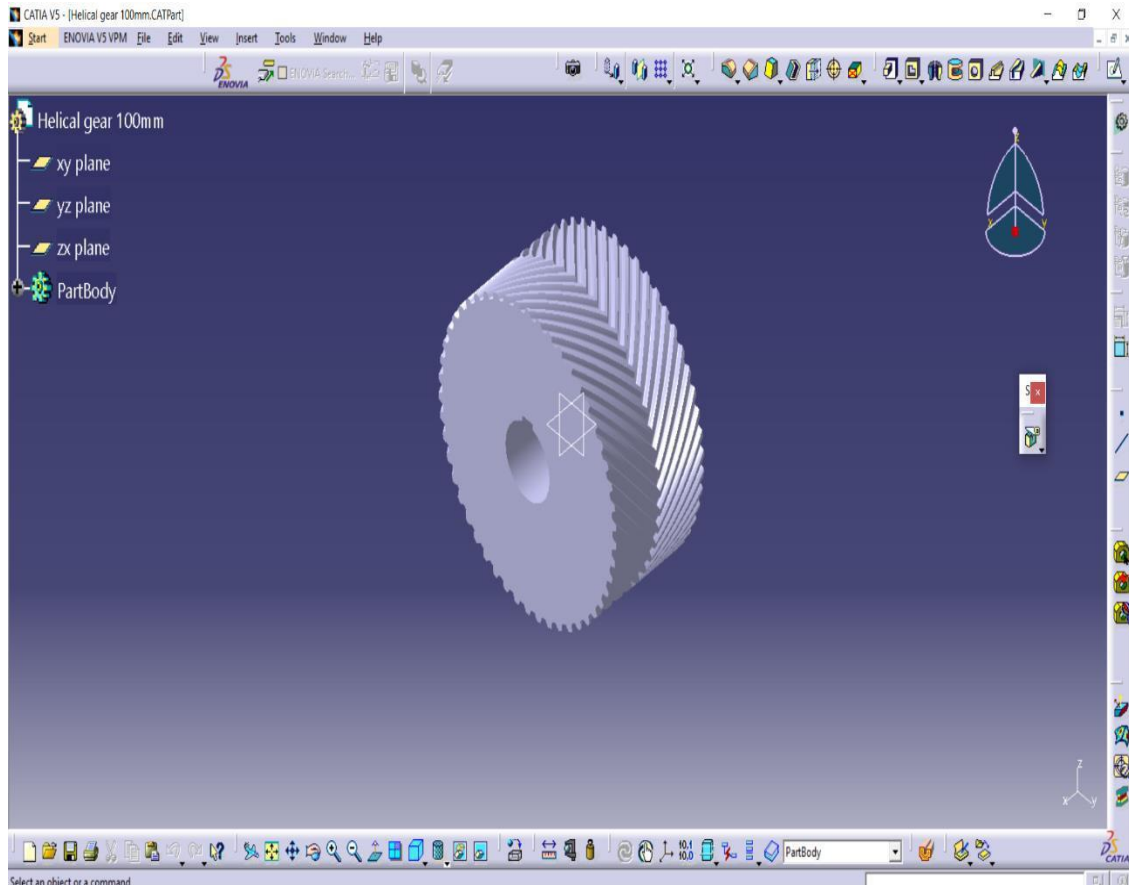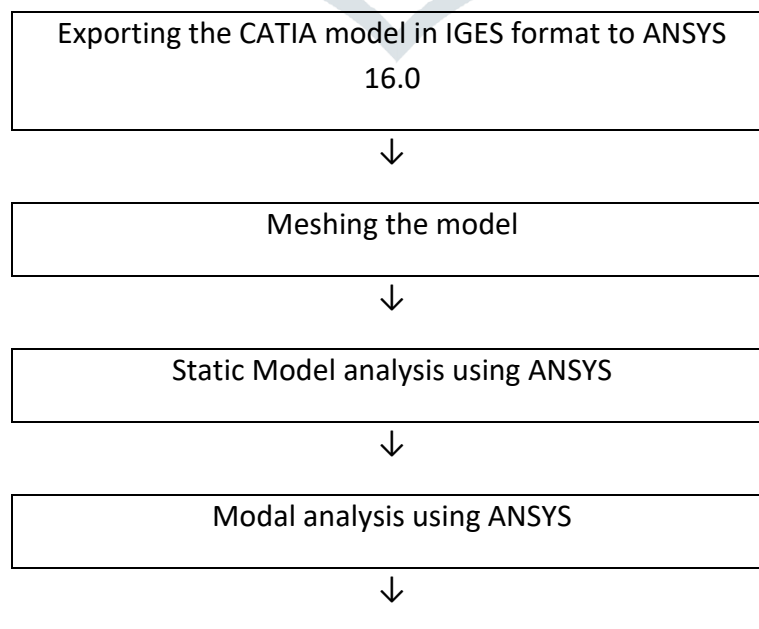

Fig. 1 Design of gear

ANSYS stands for Analysis System product. Dr. John Swanson was the founder of ANSYS Inc. In the year 1970 ANSYS was founded in order to establish a technology that facilitates several companies/industries to compute or simulate analysis issues. ANSYS is a general-purpose finite element analysis (FEA) software package that is extensively used in industries to resolve several mechanical problems. FEA is a method of fragmenting a composite system in to small pieces called elements. The ANSYS software carries out equations that regulate the performance of these elements and solves them resulting in an overall description of how system works integrally. The obtained results are displays in a tabulated or graphical form. ANSYS is a dedicated general purpose Finite Element package used for determining the temperature, stress and strains.

In this work, three-dimensional model is generated by using the CATIA V5 software, and then the model is exported to the ANSYS software in the form of IGES (Initial Graphics Exchange Specifications) file format. This provides to the ANSYS software for better understandability about the model. Here the ANSYS 16.0 Multiphysics software is used to analyze the model.
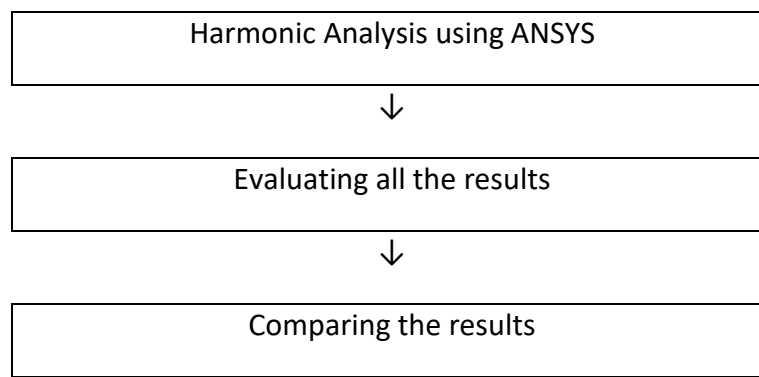
Analysis Procedure

Exporting the CATIA model in IGES format to ANSYS 16.0

↓

Meshing the model

↓

Static Model analysis using ANSYS

↓

Modal analysis using ANSYS

↓

Harmonic Analysis using ANSYS

↓

Evaluating all the results

↓

Comparing the results

Fig. 2 Flow Chart

Importing the geometry into ANSYS

The geometry which is prepared by CATIA is then imported into ANSYS workbench by opening the ANSYS software and then select any analysis to perform and add engineering data for the material properties and right click on geometry option there will be an option called import geometry. By clicking on it choose the IGES file which is already saved in a folder modelled by using CATIA .

Meshing the model

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient Multiphysics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model.
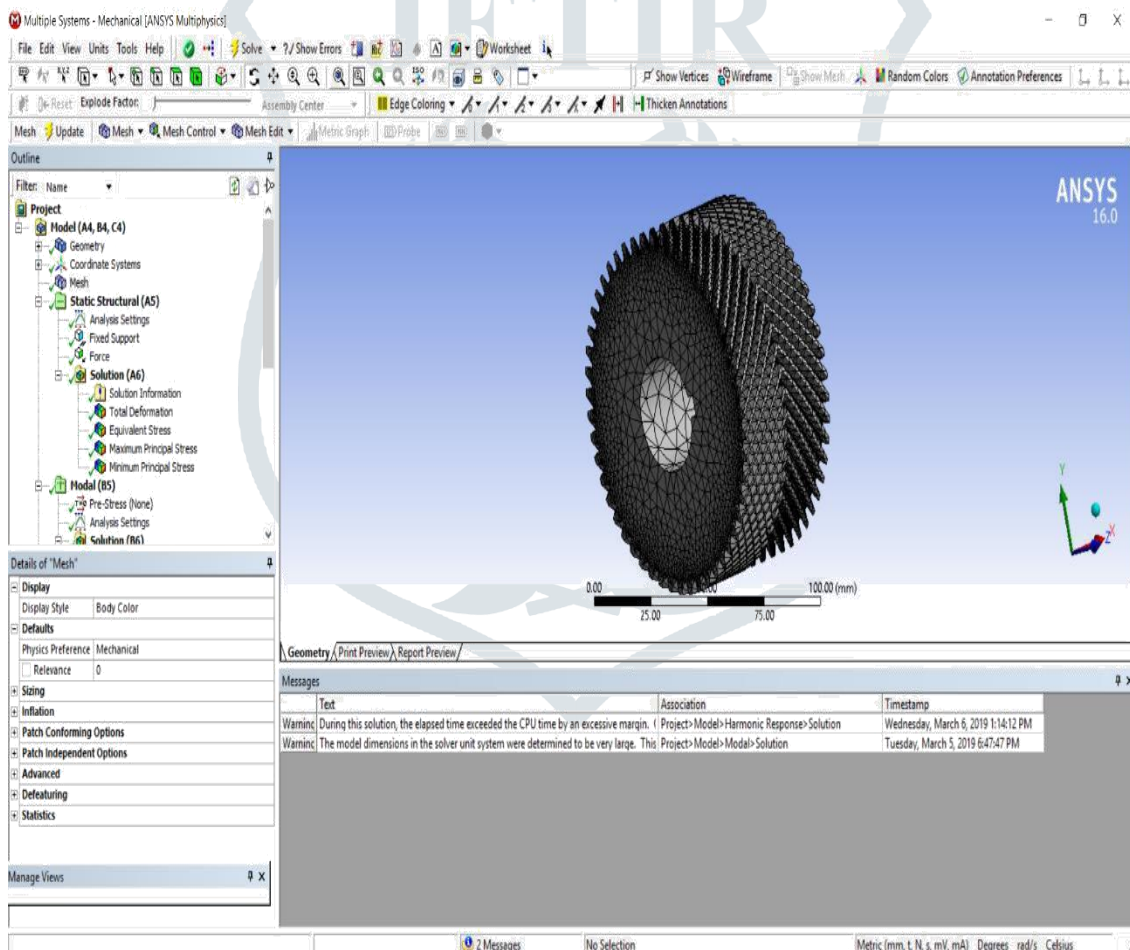

Fig. 3 Meshing the model in ANSYS

Harmonic Analysis

Harmonic analysis is to determine the steady-state sinusoidal response to sinusoidal varying loads all acting at a specified frequency. Some load types can be applied with a phase offset. Harmonic analysis, and includes a Modal analysis to characterize the structure. This Harmonic analysis is not using modal superposition, as can be seen in the fact that Solution results from the Modal analysis are not linked.

Evaluating the results

After giving all the inputs like importing the geometry, adding engineering data, fixed supports and load then the solution to be solved by inserting the required solutions like Total deformation, Equivalent stress, maximum principal stress, minimum principal stress, modal and harmonic response. By clicking on solve the results were automatically generated and the ANSYS also provides geometry, graph and report preview to easy analysis for the given model.

Comparing the results:

After evaluating all the results, the results are then compared with each other and then final conclusion is given. The results which are to be compared are Total deformation, Equivalent stress, maximum principal stress, minimum principal stress, modal and harmonic response.

Assigning Material Properties:

ANSYS software has a built-in material property that facilitates to the user easy selection of material properties. Whereas in other software this facility is not accommodated but manually create the properties to the desired material, ANSYS software has inbuilt programming for selection of material components. The below Table: 4.2 shows that the selection of material to the geometric component in ANSYS Workbench software. The material properties are density, Young's modulus, Poisson's ratio, ultimate tensile and compressive strengths .In this analysis the material properties assigned are as follows:

<div align="center">

Table 1 Materials Specifications

Gear materials	Density (kg/m³)	Young's Modulus (GPa)	Poisson's Ratio	Ultimate Tensile Strength (MPa)	Ultimate Compressive Strength (MPa)
6150 Alloy steel	7850	200	0.3	670	670
Carbon Fibre	1600	70	0.1	600	600

</div>

Von-mises stress:

The von Mises yield criterion (also known as the maximum distortion energy criterion) suggests that yielding of a ductile material begins when the second deviatoric stress invariant reaches a critical value. It is part of plasticity theory that applies best to ductile materials, such as some metals.
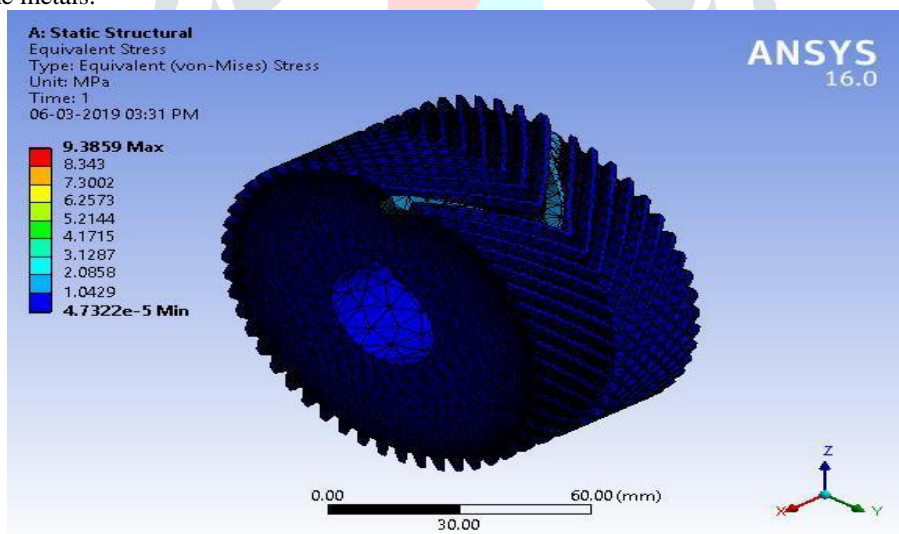

<div align="center">

Fig.4 Static Analysis of Herringbone gear

</div>

Total deformation:

Directional deformation can be put as the displacement of the system in a particular axis or user defined direction. Total deformation is the vector sum all directional displacements of the systems. The total deformation stress induced in 6150 Alloy steel after doing static structural finite element analysis is less than the deformation formed for Carbon fibre material.
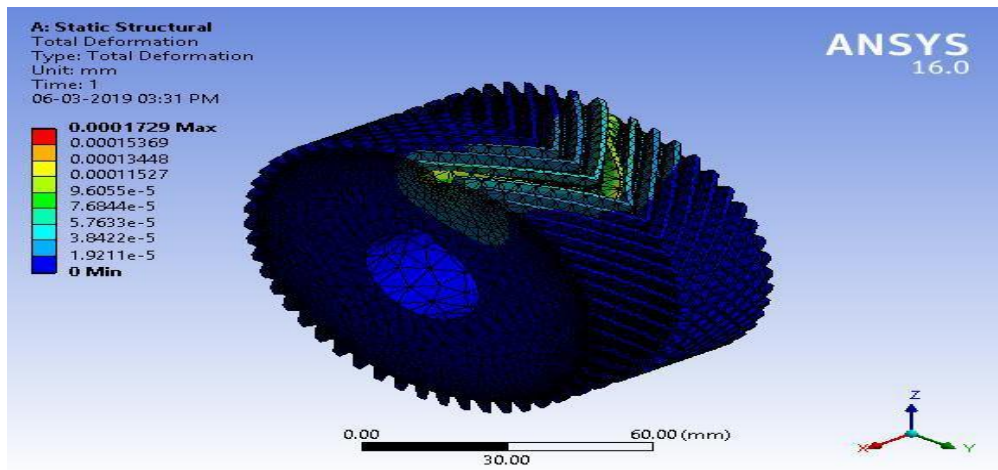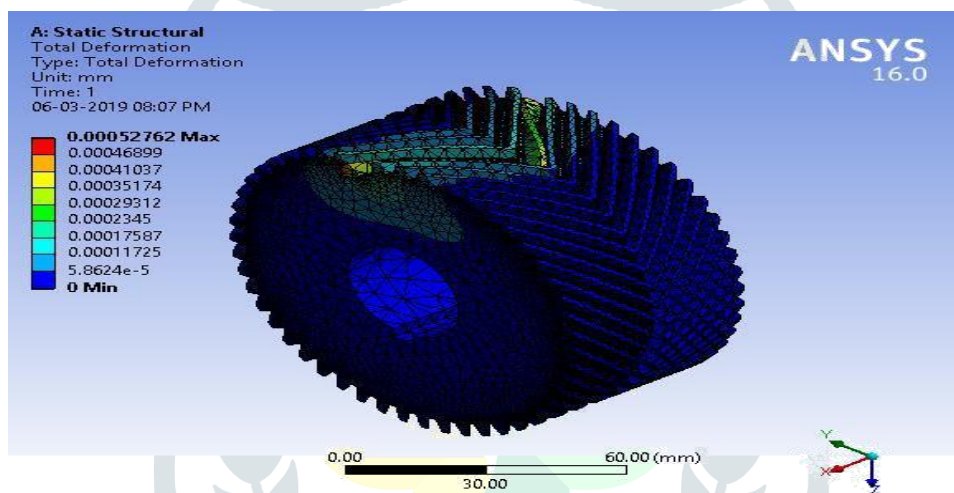

Fig.5 Total Deformation of 6150 Alloy steel


Figure.6 Total Deformation of Carbon Fibre

Modal Analysis:

Modal analysis is to determine the natural frequencies and mode shapes of a structure. Modal analysis is the study of the dynamic properties of systems in the frequency domain. A typical example would be testing structures under vibrational excitation. Modal analysis is the field of measuring or calculating and analyzing the dynamic response of structures and/or fluids or other systems during excitation.
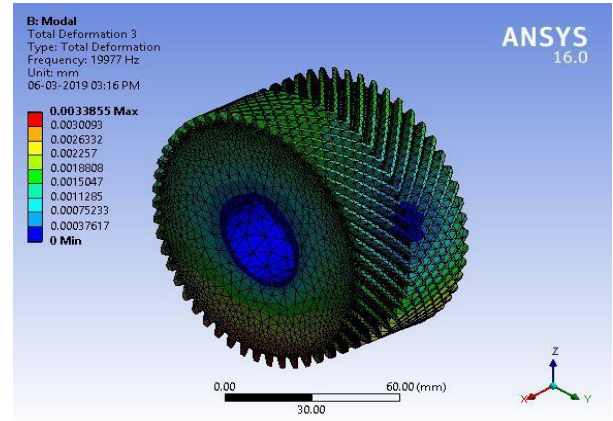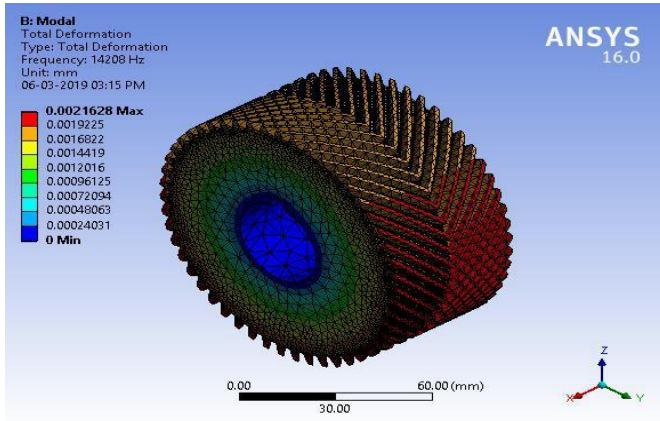
6150 Alloy Steel:
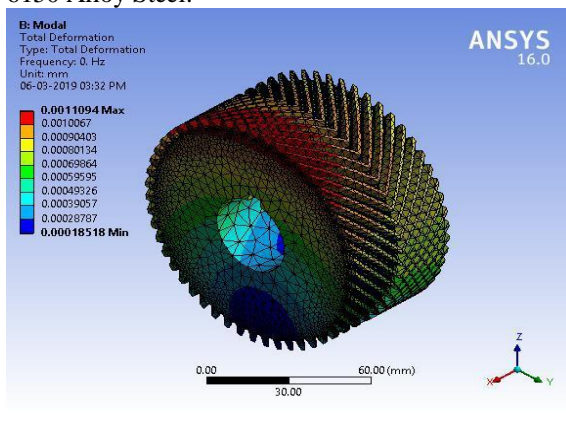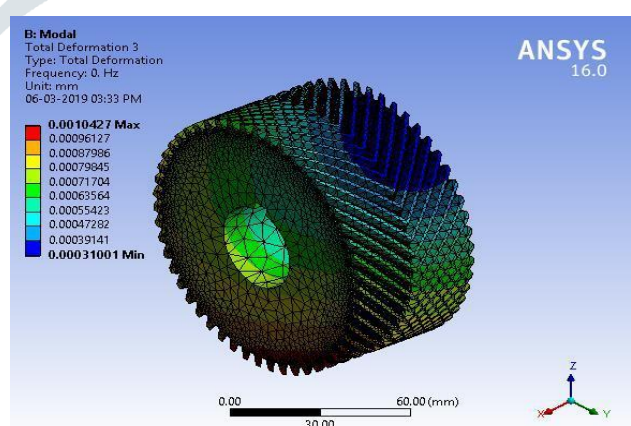
Mode-1   Mode-3

Fig.6 Modal Analysis of Alloy steel Material at Mode-1 and Mode-4

Carbon Fibre:

<center>Mode-1   Mode-3</center>

Fig.7 Modal Analysis of Carbon Fiber Material at Mode-1 and Mode-4
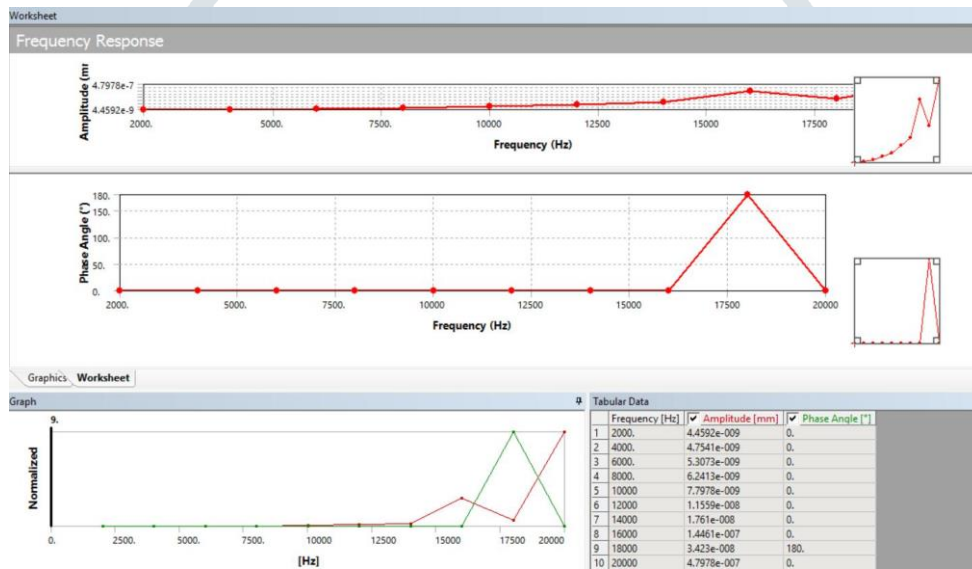
Harmonic analysis

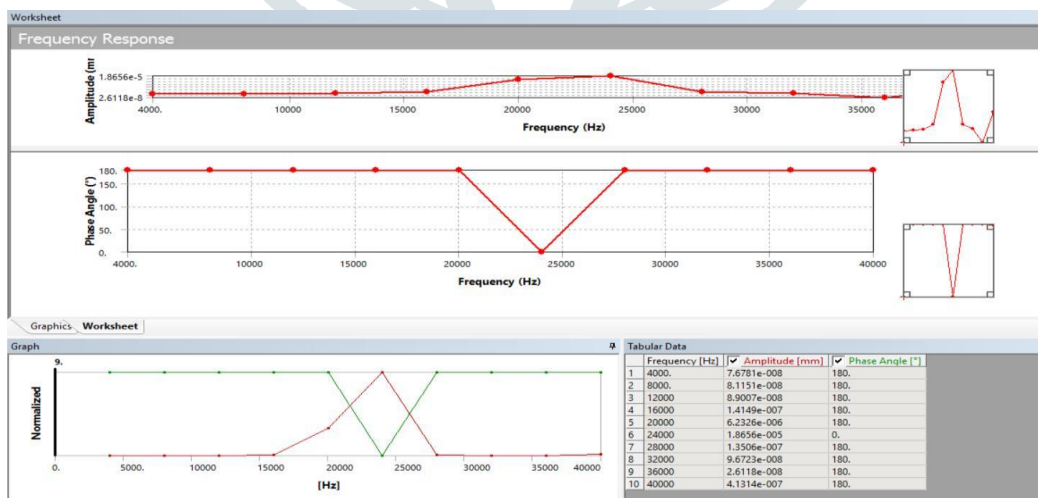**Fig. 8 Harmonic Response for 6150 Alloy Steel**

Fig.9 Harmonic Response for Carbon fiber

Amplitude vs Frequency:

The frequency is just the opposite; it's the number of wave cycles that are completed in one second. Amplitude and wavelength are both measures of distance. The amplitude measures the height of the crest of the wave from the midline. The wavelength measures the horizontal distance between cycles. The difference between frequency and amplitude is that frequency is a measurement of cycles per second, and amplitude is a measurement of how large a wave is. Amplitude represents the wave's energy. The most important terms in relation to these waves are wavelength, frequency and amplitude.
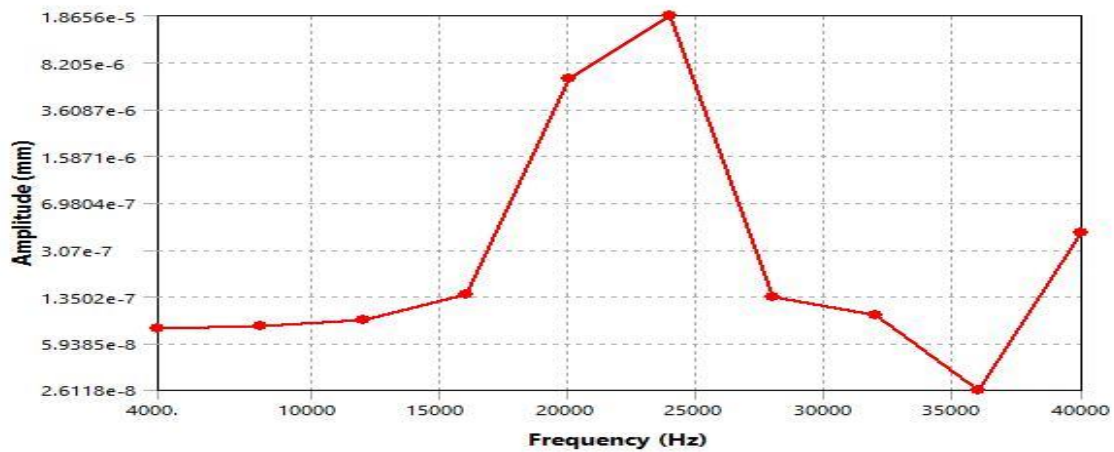

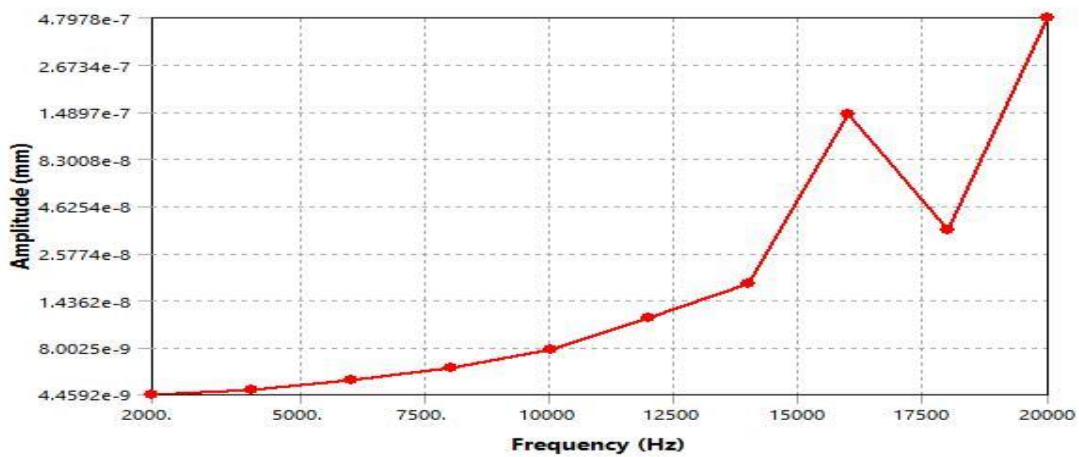
Fig.10 Frequency vs Amplitude of Carbon Fiber

Fig.11 Frequency vs Amplitude of 6150 Alloy Steel

Amplitude vs Phase angle:

Phase is not a property of just one RF signal but instead involves the relationship between two or more signals that share the same frequency. The phase involves the relationship between the position of the amplitude crests and troughs of two waveforms. Phase can be measured in distance, time, or degrees.
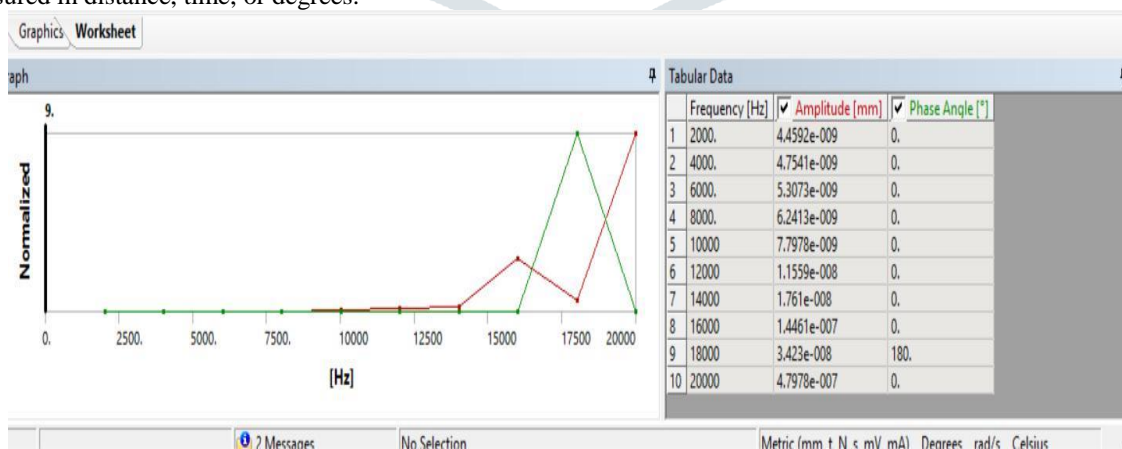

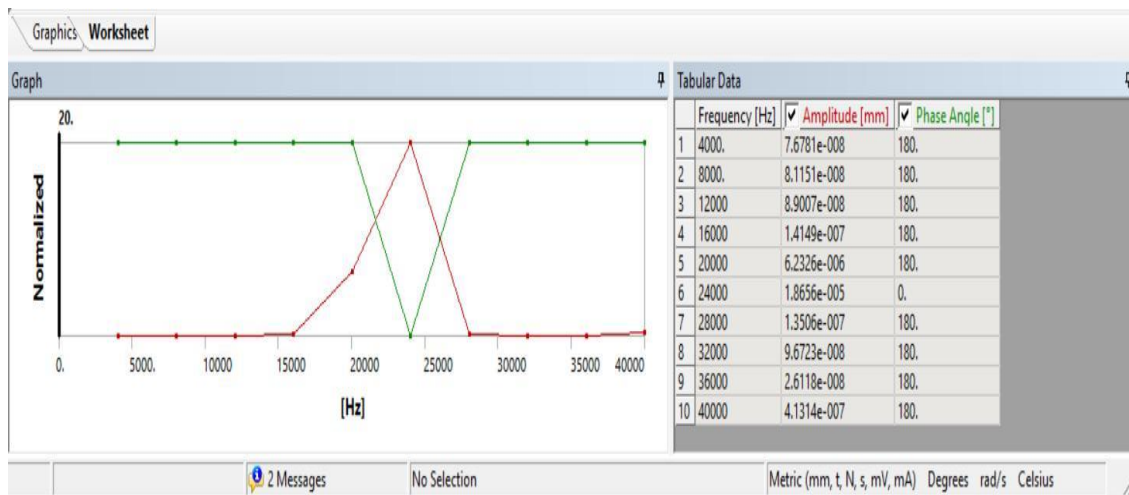**Fig.12 Amplitude vs Phase angle of 6150 Alloy Steel**

Fig.13 Amplitude vs Phase angle of Carbon Fibre

Results and Discussions

Finite element analysis of 6150 Alloy steel material after applying load and fixed constraints the maximum stress is observed. Stress distribution shown with the help of various colors. The red color indicates the maximum stress affected region, pale red and yellowish colors indicates the high stresses region but comparatively less with red color region, where as the blue color indicates the minimum stress distribution region. The maximum stress distribution showing in red color is the fracture occurring region; the minimum stress is the safest region.

Structural Analysis:

S.NO	PROPERTY	6150 Alloy Steel	Carbon Fibre
1	Deformation (mm)	0.0001729	0.00052762
2	Stress(MPa)	9.3829	8.476

Table 2 Comparison of results for Structural Analysis
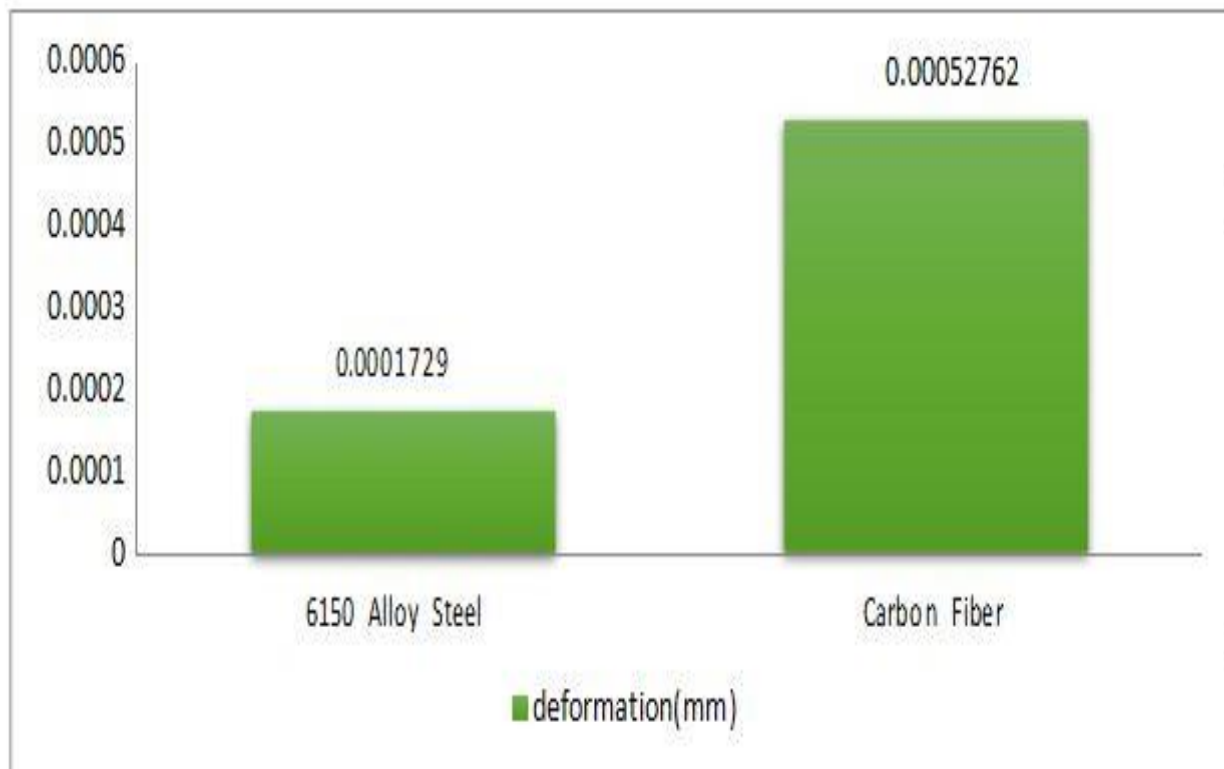
Fig.14 Deformation and Stress Variations in Static Structural Analysis

S.NO	Properties	6150 Alloy Steel	Carbon Fibre
MODE 1	Frequency(Hz)	12704	14208
	Deformation(mm)	0.0014131	0.0021628
MODE 3	Frequency(Hz)	12836	19904
	Deformation(mm)	0.001407	0.0030347
MODE 5	Frequency(Hz)	13949	19977
	Deformation(mm)	0.0008939	0.0033855

Fig.15 Comparison of Modal values

CONCLUSION

In this work, the analysis is carried out for two different materials. The Total deformation, Equivalent Stress, Minimum principal stress, Maximum principal stress, Modal analysis and Harmonic response values were calculated, graphs were plotted to analyze the results. The results were compared between two materials and discussed below.

Structural Analysis:

In structural analysis, for a given value of same load the deformation of 6150 Alloy Steel and Carbon fibre are negligible. But in deep the carbon fibre showing deformation of 67% more than Alloy Steel. But while taking stress point of view Alloy steel takes only 10.6% more than Carbon Fibre.

Modal Analysis:

In modal analysis, we performed 1-Mode, 3-Mode and 5-Mode modal analysis. In 1-Mode the deformation between two materials is very less when compared to 3-Mode and 5-Mode respectively. At 3-Mode, we have observed more deviation between 6150 Alloy Steel and Carbon fibre. While at 1-Mode and 5-Mode it is very less.

Harmonic Response:

In Harmonic analysis, we observed that in maximum and minimum frequencies between the two materials is differed by 50% but the results obtained by using Carbon fibre material shows that at higher amplitude the frequency is between 20000 to 25000 Hz, but in case of 6150 Alloy steel the frequency response at higher amplitude is at 20000Hz. Anyway both are in comparable at higher amplitudes.

Finally, concluding that Carbon fibre is performing almost 70% of the performance of 6150 Alloy Steel. So, while going economical, availability, mechanical properties like brittleness, machinability, it is advisable that Carbon fibre is one of the best substitute material after 6150 Alloy steel based on various applications of the herringbone gears.

REFERENCES

1.W. Liu, B. Tang and Y. Jiang, Status and problems of wind turbine structural health monitoring techniques in China, Renewable Energy, 35 (7) (2010) 1414–1418.

2.A Review of Planetary and Epicyclic Gear Dynamics and Vibrations, Article (PDF Available) Robert G. Parker, Christopher G. Cooley in Applied Mechanics Reviews 66(4):040804 • July 2014 with3,510 Reads DOI: 10.1115/1.4027812

3. A. Kahraman, H. N. Ozguven and D. R. Houser, "Dynamic Analysis of Geared Rotor by Finite Element", ASME Journal of Mechanical Design, 1992, Vol. 114, Issue 2, page 507- 514.

4.P. Sondkar and A. Kahraman, A dynamic model of a double-helical planetary gear set, Mechanism and Machine Theory, 70 (2013) 157–174.

5.Bu, Geng Liu, Liyan Wu; "Modal analyses of herringbone planetary gear train with journal bearings Mechanism and Machine Theory", Volume 54, August 2012, Pages 99–115

6..Design and analysis of herringbone gear with sixth-order transmission error based on meshing vibration optimization Article in Advances in Mechanical Engineering 9(6):168781401770435 • June 2017 with400 Reads DOI: 10.1177/1687814017704359