Model Analysis of Leaf Spring Made of Composite Material

Dr. M. Easwaramoorthi[#1], A. Pathri [#2], V. S. Prasanth[#3], S. Rajagopal[#4], k. Sridhar[#5].

[#1]*Assistent professor , Department of Mechanical Engineering, Nandha Engineering College, Erode-638052. India.*
[#2,3,4,5]*UG scholor , Department of Mechanical Engineering, Nandha Engineering College, Erode-638052. India.*

Abstract— **Leaf springs are one of the oldest suspension components they are still frequently used, especially in commercial vehicles. The past literature survey shows that leaf springs are designed as generalized force elements where the position, velocity and orientation of the axle mounting gives the reaction forces in the chassis attachment positions. Another part has to be focused, is the automobile industry has shown increased interest in the replacement of steel spring with composite leaf spring due to high strength to weight ratio. Therefore, analysis of the composite material becomes equally important to study the behavior of Composite Leaf Spring. The objective of this paper is to present modeling and analysis of composite mono leaf spring (GFRP) and compare its results. Modeling is done using solid works 2015and Analysis is carried out by using ANSYS 15.0 software for better understanding.**

Keywords—**Leaf Spring, COMPOSITE, FEA, ANSYS.**

INTRODUCTION

The purpose of this final year project is to analyses leaf spring failure. The client for this project is the R&D department of Normal Group. The spring is used in the RBO underground mining machine, manufactured by Normal Corporation.

The main objective is to gather information about the effect of the improvement on the spring structure. The secondary objective is to get information about the phenomena which have effect on the fracture of the spring. The assumption at the start of the project is that the spring takes quite a high twisting torque around longitudinal direction. The twisting torque is generated from inertial forces of the vehicle body mass.

Figure 1.1. Broken spring in the front axle.

The situation at the beginning of this project is that the life time of a single spring is too low. The duration of one spring, according to customers' report, was from 100 hours to few months. The most common fracture point was located close to the spring eye in the fixed support side where the profile is thinnest. The broken spring is presented in Figure

The analysis will be carried out by using different kinds of approach methods. The process flow of the analysis is presented in Figure 3. The project was divided into theoretical and experimental sections. The theoretical phase consists of the multi-body system(MBS) and the finite element (FE) approach.

Figure 1.2. Spring with the clamp installed. (Normet Group, Marketing databank)

The experimental phase consists of series of strain gage measurements to analyses stress levels and to predict the life cycle of the spring. The basic idea is to collect data from different outputs to verify and compare the results.

2. LITERATURE SURVEY

Santosh Krishnaji [1] Research studied in this leaf spring is widely used in automobiles and one of the components of suspension system. It needs to have excellent fatigue life. As a general rule, the leaf spring must be regarded as a safety component as failure could lead to severe accidents. The purpose of this paper is to analyze static, modal and predict the fatigue life of steel leaf spring along with analytical stress and deflection calculations. This present work describes static, modal and fatigue analysis of a existing leaf spring and modifying existing steel leaf spring by reducing no of graduated leaves and increasing thickness. The dimensions of a modified leaf spring of a LCV are taken and are verified by design calculations. The non-linear static analysis of 2Dmodel of the leaf spring is performed using NASTRAN solver and compared with analytical results. The fatigue life is carried out by MS fatigue. The pre processing of the modified model is done by using HYPERMESH software.

Ms. Sarika [2] considered FEA analytical calculation - Leaf springs are one of the oldest suspension components they are still frequently used, especially in commercial vehicles. Weight reduction is now the main issue in automobile industries. In the present work, existing mono steel leaf spring of a Tata Bus is taken for modeling and analysis. Finite element method has been implemented to modify the existing leaf spring with considering the dynamic loading. This work involves design and analysis of a conventional leaf spring under static and dynamic loading conditions. The 3D model is prepared in Creo 1.0, and then analysis is performed in the ANSYS 11.0 by considering same load in static and dynamic loading. For the cost reduction in existing leaf spring modification carried out by iteration method considering three cases such as varying number of leaves, varying width and varying thickness. The optimization has been carried out to satisfy the permissible value of factor of safety. The results are verified by comparison of Analytical and Finite Element Method. All analytically calculated values of deflection and stresses are closely matching with values obtained from ANSYS software.

G. Goudah [3] An automotive suspension system is designed to provide both safety and comfort for the occupants. When a vehicle encounters a road surface irregularity, the tire deforms and the suspension displaces. The result of such disturbance will cause some energy lost which will be dissipated in the tires and the shock absorber while the remainder of the energy is stored in the coil spring. In this paper, Finite element models were developed to optimize the material and geometry of the composite elliptical spring based on the spring rate, log life and shear stress. The influence of elasticity ratio on performance of woven roving wrapped composite elliptical springs was investigated both experimentally and numerically, the study demonstrated that composites elliptical spring can be used for light and heavy trucks with substantial weight saving. The results showed that the elasticity ratio significantly influenced the design parameters. Composite elliptic spring with elasticity ratios of a/b= 2 displayed the optimum spring model.

Zhao Yongjuan [4] Based on the request for the reliability design of electric shoot primer automatic filling device, use ALGOR software to conduct modeling of the leaf spring that connects firing pin and electric shoot device, carry on displacement deprogram and equivalent deprogram analysis on parameters such as leaf spring hole depth, leaf spring thickness, etc , and design a leaf spring with long life and reliable working, which realizes electromechanical dual-purpose shoot device of a gun.

Amol Bhanage [5] In the present work, steel leaf spring used in passenger cars is replaced with a composite leaf spring made of a glass/epoxy composite The primary objective is to compare fatigue characteristics of SAE1045-450-QT steel and E - Glass/ Epoxy Composite material. Based on the available design data a fatigue analysis is carried out on an ANSYS Workbench v14.0 and the results of the simulation are documented. Factors like fatigue life, fatigue damage, biaxiality indication, rain flow counting and fatigue response are plotted for the composite leaf spring and the fatigue performance is predicted using life data. Therefore the objective of this paper is to present a design and simulation study on the fatigue performance of a glass fiber/epoxy composite leaf spring through design and finite element method and prove the reliability of the validation methods based only on simulation, thereby saving time, material and production costs for a complete product realization.

Y.Venu [6] The objective of this present work is to estimate the deflection, stress and mode frequency induced in the leaf spring of an army jeep. The emphasis in this project is on the application of computer aided analysis using finite element concept. The component chosen for analysis is a leaf spring which is an automotive component used to absorb vibrations induced during the motion of vehicle. It also acts as a structure to support vertical loading due to the weight of the vehicle and payload. Under operating conditions, the behavior of the leaf spring complicated due to its clamping effects and interleaf contact, hence its analysis is essential to predict the displacement, mode frequency and stresses. The leaf spring, which we are analyzing, is a custom designed leaf spring with different eyes like viz. Berlin and upturned eyes with different materials at different sections. This spring is intended to bear heavy jerks and vibrations reduced during real operating conditions in military operations. In analysis part the finite element of leaf spring is modeled using solid tetrahedron10-NODE-187 elements. Appropriate boundary conditions, material properties and loads are applied selected as per intended performance. The resultant deformation, mode frequencies and stresses obtained are analyzed.

Narendra Yadav [7] A leaf spring is simple form of spring, generally used for the suspension in automotives. Earlier it was like a slender arc-shaped having length of a spring steel of rectangular cross section. In this paper analysis is done for leaf spring whose thickness varies from the center to the outer side following a parabolic pattern. The development of a parabolic tapered leaf spring enabled the springs to become lighter, but also provides a much improved ride to the vehicle through a reduction on interleaf friction. To move further, authors take an opportunity to perform a Finite element analysis (FEA) on the spring model so that stress and damage distribution can be observed. In this paper, initially the magnitude of stress pertaining to parabolic leaf spring is computed by finite element method and then the approach to minimizing the stress has been carried out effectively with help of Local Algorithm for Constants and Priorities.

W. Hufenbach [8] Springs within the suspension of vehicles are commonly designed for maximum load. Thus, the driving dynamics of the unloaded or lightly loaded vehicle is poor. Furthermore, the high stiffness of the spring leads to dynamically caused peak loads on the structure of the vehicle To avoid these problems, springs with adjustable spring rates are necessary. Especially in trucks, air springs are common, where air pressure is used to adjust the spring rate. However, these air springs are high maintenance and more expensive compared to classic metallic leaf springs. This paper presents a new concept to adjust the spring rate of a light weight spring system containing glass fiber-reinforced polypropylene (GF/PP) leaf spring elements.

Chang Keng Fuh [9] Taguchi robust design is an important engineering methodology which renders a product or a process insensitive to the effects of variability y and improves the performance at low cost including in static and dynamic characteristics. Dynamic characteristic is also known as the "Signal-Response System". Dynamic characteristics of an experimental design can be implemented by extending the static characteristics of an experiment with another signal level. Besides that, it involves a two steps optimization procedure, in which initially the variation around a linear fact ion is minimized, and secondly the sensitivity of the linear function is adjusted to a target value. The goal in a dynamic characteristics experiment is to find the control factors that make the response least sensitive to noise and also the control factors that will give a unit gradient between different signal and the response. In this research, the semi-elliptical leaf spring equation is used to demonstrate the principal of dynamic characteristics by using computer simulation. The deflection of the semielliptical leaf spring is optimized over a range of applied load. The sensitivity of the signal-response relationship is adjusted to a target value by an appropriate setting of control factors and their levels. Lastly, control factors that largely affect the sensitivit y and variability y will be selected as the optimum condition. The result will show an improvement in quality y loss after the optimization.

Meghavath. Peerunaik [10] The objective of this present work is to estimate the deflection, stress and mode frequency induced in the leaf spring of an army jeep design by the ordinance factory. The emphasis in this project is on the application of computer aided analysis using finite element concept. The component chosen for analysis is a leaf spring which is an automotive component used to absorb vibrations induced during the motion of vehicle. It also acts as a structure to support vertical loading due to the weight of the vehicle and payload. Under operating conditions, the behavior of the leaf spring s complicated due to its clamping effects and interleaf contact, hence its analysis s essential to predict the displacement, mode frequency and stresses. The leaf spring, which we are analyzing, is a specially designed leaf spring used in military jeeps. This spring is intended to bare heavy jerks and vibrations reduced during military operations. A model of such jeep has been shown in this project report. In analysis part the finite element of leaf spring is created using solid tetrahedron elements, appropriate boundary conditions are applied, material properties are given and loads are applied as per its design, the resultant deformation, mode frequencies and stresses obtained are reported and discussed.

Veeramalai Chinnasamy le atal.,[11] In an automotive system, a curved leaf spring is used for the purpose of suspension and for reducing the transient vibration of the system. Composite materials are widely used in automobile industries as a replacement for steel to reduce the weight and to increase the strength of an automotive system. In this study, various materials have been considered for an analysis based on the Young modulus-to-yield strength ratio. The study has been carried out by considering the material properties. The contact analysis is performed with a curved beam of three leaves which is an equivalent to a semi-elliptical leaf spring used for an automotive suspension system. A comparison between analytical and simulation results shows that the material properties, such as the Young modulus and the yield strength, are very important in the design and development of a composite leaf spring. The composite material is compared with other materials, with the former showing good suspension function and better reliability.

Modeleepak Kumar [12] The objective of this script is to automate the Finite Element Modeling of Leaf Spring for its Torsional Stiffness and Lateral Stiffness validation. This is essential in the process of MBD model building of Commercial Vehicles for extracting attachment point Forces and Moments to validate the Frame Design through FEA. This script is designed to use with Altair Hyper Mesh v11.0 environment. Each leaf is modeled and assembled with the others according to the input parameters such as the Leaf Length, Leaf Thickness, Leaf Width, Centre Hole Diameter, etc with the required mesh quality. Assembly is completed by proper bolting at the centre and clamping using the clamps according to the Clamp Height, Thickness, Width and Distance from Centre. Proper Material and Section Properties are defined for each Part. Contact is defined between each leaf. Bolt pretension is defined at the centre. Boundary Conditions and Loads are defined for the required Load Cases. Output Requests for FEA are defined. Script renumbers the required nodes for the ease of Post Processing. It creates IGES / STEP CAD File from FE Model for Reference Purpose. This script reduces the manual effort and eliminates manual errors.

M. M. Patunkar [13] Leaf springs are one of the oldest suspension components they are still frequently used, especially in commercial vehicles. The past literature survey shows that leaf springs are designed as generalized force elements where the position, velocity and orientation of the axle mounting gives the reaction forces in the chassis attachment positions. Another part has to be focused, is the automobile industry has shown increased interest in the replacement of steel spring with composite leaf spring due to high strength to weight ratio. Therefore, analysis of the composite material becomes equally important to study the behavior of Composite Leaf Spring. The objective of this paper is to present modeling and analysis of composite mono leaf spring (GFRP) and compare its results. Modeling is done using Pro-E (Wild Fire) 5.0 and Analysis is carried out by using ANSYS 10.0 software for better understanding.

Mehdi Bakhshesh [14] Springs that can reserve high level of potential energy, have undeniable role in industries. Helical spring is the most common element that has been used in car suspension system. In this research, steel helical spring related to light vehicle suspension system under the effect of a uniform loading has been studied and finite element analysis has been compared with analytical solution. Afterwards, steel spring has been replaced by three different composite helical springs including E-glass/Epoxy, Carbon/Epoxy and Kevlar/Epoxy. Spring weight, maximum stress and deflection have been compared with steel helical spring and factors of safety under the effect of applied loads have been calculated.

4. METHODOLOGY

3.1. PLANNING:

It is the process of thinking about and organizing the activities required to achieve a desired goal. Planning involves the creation and maintenance of a plan.

3.2. BLOCK DIAGRAM:

In this process the idea of the project is converted as the rough diagram, by the required drawing tools.

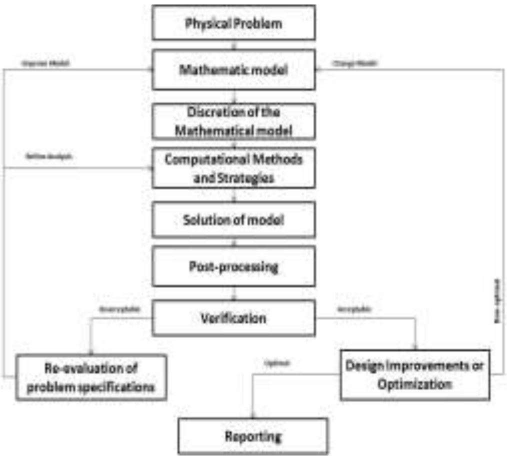

Figure 3.1 Methodology flowchart

3.3. DESIGN:

Design is the creation of a plan or convention for the construction of an object. It is strategic approach for someone to achieve a unique expectation. It defines the specifications, planes, activities and process economic constraints in achieving the objective.

3.4. SELECTION OF MATERIAL:

In this process to identify the material for make a prototype (or) product. The material selection is the depends upon the application. So the material is should be in applicable one and less cost.

3.5. MACHINING:

In this process the materials are to be machined for required dimension and then to obtain the required surface finish.

3.6. TESTING: Testing is the process of evaluating a system or its component(s) with the intent to find whether it satisfies the specified requirements or not. Testing is executing a system in order to identify any gaps, errors, or missing requirements in contrary to the actual requirements.

3.7. CHECKING:

The act or an instance of inspecting or testing of the project. Sometimes we assume something is correct without making sure. Checked out the system to make sure there were no errors in the software. The checking is satisfied the next step will be processed. But it not satisfied the design is changed.

5. PROBLEM IDENTIFICATION

Leaf spring is one of the most critical components of internal combustion (IC) engines bearing the statically and dynamically fluctuating loads .The optimization of Leaf spring had already started as long back. However, everyday consumers are looking for the best from the best. That's why optimization is really important for the automotive industry especially. Optimization of the component is to reduce time for the production of the product that is stronger, lighter and that has less total cost during production. The design and weight of the Leaf spring influence on the car's performance. Hence, it affects the car manufacture's credibility. Change in the design and material results a significant decrement in weight and an increment in the performance of the engine. The structural factors considered for weight reduction during the optimization include the buckling load factor, stresses under the loads, bending stiffness, and axial stiffness.

The benefits of Leaf spring optimization are eventually gone back to consumer itself. Among the main objectives are to improves the engine performance and also to strengthen the product that is ensure the safety of human being. By maximize the strength automatically it will longer the life cycles of the Leaf spring. Lots of knowledge will be apply and produced during the process. In this study, the design of the Leaf spring will be improve and at the same time increase the strength. The study will be focus on the finite element modeling and analysis. From the analysis results, the decision whether Leaf spring needs to be redesign or not will based on the topology optimization. Testing is the process of evaluating a system or its component(s) with the intent to find whether it satisfies the specified requirements or not. Testing is

6. MATERIALS USED

The Leaf spring are most usually made of Forged steel or En8 for production engines. These materials have different properties and suitable for different engines. But in this project Automotive Alumina is used as Leaf spring material.

7. EN8 STEEL

BS970: 1955 EN8, BS970/PD970: 1970 onwards 080M40

EN8 is usually supplied untreated but can be supplied to order in the normalized or finally heat treated (quenched and tempered to "Q" or "R"properties for limiting ruling sections up to 63mm), which is adequate for a wide range of applications. Please refer to our selection guide for comparisons.

EN8 is a very popular grade of through-hardening medium carbon steel, which is readily machinable in any condition. (Refer to our machinability guide). EN8 is suitable for the manufacture of parts such as general-purpose axles and shafts, gears, bolts and studs. It can be further surface-hardened typically to 50-55 HRC by induction processes, producing components with enhanced wear resistance. For such applications the use of EN8D (080A42) is advisable. It is also available in a free-machining version, EN8M (212A42).

EN8 in its heat treated forms possesses good homogenous metallurgical structures, giving consistent machining properties. Good heat treatment results on sections larger than 63mm may still be achievable, but it should be noted that a fall-off in mechanical properties would be apparent approaching the centre of the bar

080M40 (EN8) - mechanical properties		
Max Stress	700-850 n/mm^2	
Yield Stress	465 n/mm^2 Min	(up to 19mm LRS)
0.2% Proof Stress	450 n/mm^2 Min	(up to 19mm LRS)
Elongation	16% Min	(12% if cold drawn)
Impact KCV	28 Joules Min	(up to 19mm LRS
Hardness	201-255 Brinell	

It is therefore recommended that larger sizes of EN8 are supplied in the untreated condition, and that any heat treatment is carried out after initial stock removal. This should achieve better mechanical properties towards the core.

Table.7.1 Mechanical properties

EN8 Equivalents	
BS970: 1955	EN8
BS970/PD970: 1970 onwards	080M40
European	C40, C45, Ck40,Ck45, Cm40, Cm45
Werkstoff No.	1.0511, 1.1186, 1.1189
US SAE (AISI)	1039, 1040, 1042, 1043, 1045

7.2. Automotive aluminium

Aluminium (or aluminium) is a chemical element in the boron group with symbol Al and atomic number 13. It is silvery white, and it is not soluble in water under normal circumstances. Aluminium is the third most abundant element (after oxygen and silicon), and the most abundant metal, in the Earth's crust. It makes up about 8% by weight of the Earth's solid surface. Aluminium metal is so chemically reactive that native specimens are rare and limited to extreme reducing environments. Instead, it is found combined in over 270 different minerals. The chief ore of aluminium is bauxite. Aluminium is remarkable for the metal's low density and for its ability to resist corrosion due to the phenomenon of passivation. Structural components made from aluminium and its alloys are vital to the aerospace industry and are important in other areas of transportation and structural materials. The most useful compounds of aluminium, at least on a weight basis, are the oxides and sulphates.

Aluminium alloys are alloys in which aluminium (Al) is the predominant metal. The typical alloying elements are copper, magnesium, manganese, silicon and zinc. There are two principal classifications, namely casting alloys and wrought alloys, both of which are further subdivided into the categories heat-treatable and non-heat-treatable. About 85% of aluminium is used for wrought products, for example rolled plate, foils and extrusions. Cast aluminium alloys yield cost effective products due to the low melting point, although they generally have lower tensile strengths than wrought alloys. The most important cast aluminium alloy system is Al-Si, where the high levels of silicon (4.0% to 13%) contribute to give good casting characteristics. Aluminium alloys are widely used in engineering structures and components where light weight or corrosion resistance is required.

7.2.1 Automotive Alumina Composition

Figure 7.2 Automotive Alumina Material

GENTRATIVE THE DESIGN AND DEVELEOPMENT

8. INTRODUCTION ABOUT SOLID WORKS

In Part modelling you can create a part from a conceptual sketch through solid feature-based modelling, as well as build and modify parts through direct and intuitive graphical manipulation.

The Part Modelling Help introduces you to the terminology, basic design concepts, and procedures that you must know before you start building a part. Part Modelling shows you how to draft a 2D conceptual layout, create precise geometry using basic geometric entities, and dimension and constrain your geometry. You can learn how to build a 3D parametric part from a 2D sketch by combining basic and advanced features, such as extrusions, sweeps, cuts, holes, slots, and rounds.

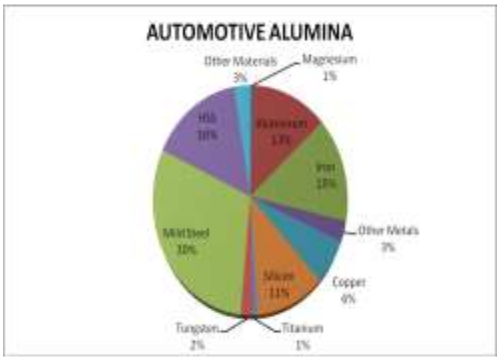

Figure.7.3 Automotive Alumina Composition

Finally, Part Modelling Help provides procedures for modifying part features and resolving failures. From the above modelling concept we modelled the component and the representation of the crutches assembly is as follows.

8.1 Three-Dimensional Modelling

Displays 3-D impact force field concentration distribution in profile with material section flow field. It can be seen that impact from varies human weight to be analysis basis to performing with respect to non uniform flow analysis to import to the model.

8.2 Design Concepts For Creating Leaf Spring

You can design many different types of models in Solid works. However, before you begin your design project, you need to understand a few basic design concepts:

1.Design Intent— before you design your model, you need to identify the design intent. Design intent defines the purpose and function of the finished product based on product specifications or requirements. Capturing design intent builds value and longevity into your products. This key concept is at the core of the Solid works feature-based modeling process.

2.Feature-Based Modeling—Solid works part modeling begins with creating individual geometric features one after another. These features become interrelated to other features as you reference them during the design process.

3.Parametric Design— The interrelationships between features allow the model to become parametric. So, if you alter one feature and that change directly affects other related (dependent) features, then Solid works dynamically changes those related features. This parametric ability maintains the integrity of the part and preserves your design intent.

4.Associability—Solid works maintains design intent outside Part mode through associatively. As you continue to design the model, you can add parts, assemblies, drawings, and other associated objects, such as piping, sheet metal, or electrical wiring. All of these functions are fully associative within Solid works. So, if you change your design at any level, your project will dynamically reflect the changes at all levels, preserving design intent.

Solid works Part enables you to design models as solids in a progressive three-dimensional solid modeling environment. Solid models are geometric models that offer mass properties such as volume, surface area, and inertia. If you manipulate any model, the 3-D model remains solid.

Figure 8.1 Leaf spring models

Solid works provides a progressive environment in which you create and change your models through direct graphical manipulation. You drive the design process for your project by selecting an object (geometry) and then choose a tool to invoke an action on that object. This object-action workflow provides greater control over the design of your models while allowing you to express your creativity. The user interface provides further support for this design process

As you work with your model, the context sensitive user interface guides you through the design process. After you choose an object and an action, Solid works interprets the current modeling context and presents requirements and optional items to complete the task. This information is displayed in a non obtrusive user interface called the dashboard that enhances your ability to directly work with your models by assessing your actions and guiding you through the design process.

The Solid works progressive modeling environment streamlines the design process enabling you to concentrate on product development and drive your designs to new levels of creativity.

9.3 FE MODEL OF THE SPRING

The FE-model was created from the 3D model of the spring. Modeling was done by using the Autodesk Inventor 2010 CAD program. The analysis was done with the Ansys Workbench 13.0. The assembly was determined with components which were essential for calculations. Interesting points for the analysis were the thinnest cross section areas in the spring profile. The 3D model is presented in Figure 16. Simplifications of the model were that elasticity of flexible bushings (3. in Figure 16) and rubber mount (6. in Figure 16) are not taken into account in the FE model. The contact friction between the rubber mount and the main spring is not modelled. Boundary conditions are modelled into the spring's eyes by using the cylindrical and remote displacement supports. Supports and the global coordinate system are introduced in Figure 17. The cylindrical support in the point A allows rotation over the z-axis with respect to the global coordinate system. The remote displacement allows rotation over the z-axis and displacement along the x-axis in the global coordinate system.

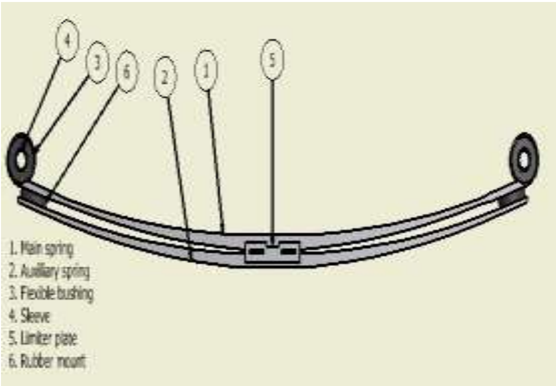

Figure 9.1.3D-model of the spring.

The mesh was generated by using the Ansys workspace automatic tool. The tool generated automatic elements to solid bodies. The mesh was refined in examination and contact areas. Refining was also done for flanks of the spring. The meshed model of the spring is shown in Figure 18. Model contained 303256 nodes and 183984 elements.

A

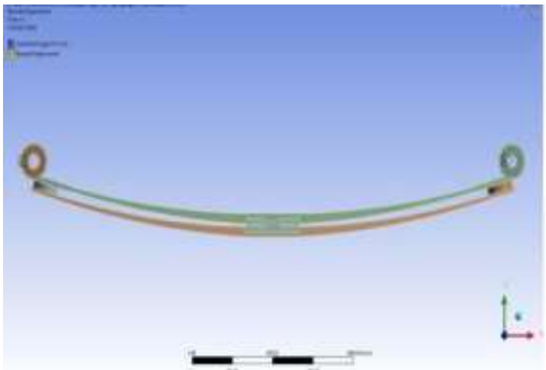

Figure 9.2 Supports of the spring.

Contacts of the model are bounded except for the contacts between rubber mounts and the main spring which are determined as frictionless connections. This type of connection describes the behavior of the spring in an ideal state where the friction between leaves is not wanted. Contact settings of the frictionless connection are presented in Figure 19. The formulation type in the nonlinear contact was set to the augmented Lagrange. This is usually more appropriate for the nonlinear analysis. The interface treatment is set to be in 'Adjust to Touch' which ignores any initial penetration and creates the stress free state to contact surfaces. The time step control uses automatic bisection. This setting evaluates the contact behavior at the end of each sub step. If penetration or drastic changes are detected the sub step is evaluated again with the time increment which is reduced by half.

9.4 IMPORTING A GEOMETRY

In this prompt, the dimension of the real part model has been measured to design a modelling approach the related journal in the drawings using offline process. Using this process, the real part model can be imported in CAD modelling as *.prt file. Thus the imported *.prt file can be viewed by the modelling software i.e solid works version 2012.

In solid works, the 3D dimension of the imported part model has been converted into *.parasolid file. Now we can be able to set the actual dimension, appearances for the converted model file. After setting the require data in solid works, the file can be imported to the analysis software i.e., ANSYS WORKBENCH. This part model may be imported in to ansys as *.IGES file for the purpose of analyzing using solid works. Thus the IGES file has been imported in ansys work bench.

In ansys workbench, the thermal transient analysis in varies cross section analysis has been made on the IGES file. After the analysis process has been completed, the data's can been stored in workbench. Then it can be viewed in ansys workbench product launcher as a link to ansys product launcher. Thus the result can be generated in the general post processor using the ansys product launcher.Now we can generate the plot Results, result summary and fluid thermal stress values advanced crutches analysis using impact values it easily assigned. The impact forces stress has been calculated. Finally the calculated output results from ansys workbench can be compared with the three dimensional discretization model done using FEA.

B

The suspension s models are analyzed using ANSYS Workbench Version 14.5 as outlined in Section **Error! Reference source not found.**. The models are meshed using the ANSYS default settings for path conforming volume with a relevance of 0.5 on a scale of -100 to 100. The resulting mesh is shown in Figure .

Figure 9.3: Default ANSYS volume Mesh

Similar setting the mesh relevance to a maximum of 100 refines the mesh to 185237 nodes and 95422 elements, an almost three-fold increase in number of mesh nodes. The refined mesh is shown in **Error! Reference source not found.**. Introducing different position of spring analysis is considering it.

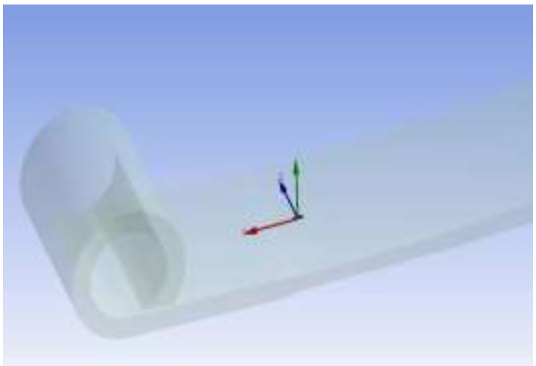

Figure 9.4 Location of the upper strain gage rosette.

Result and Discussion

1.Stress vs. super composite materials

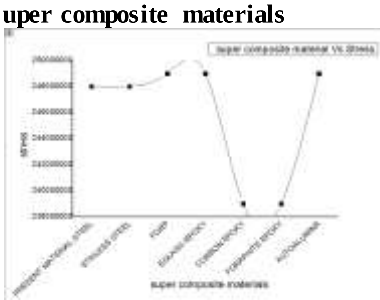

figure 10.1 flat spring stress VS. super composite materials

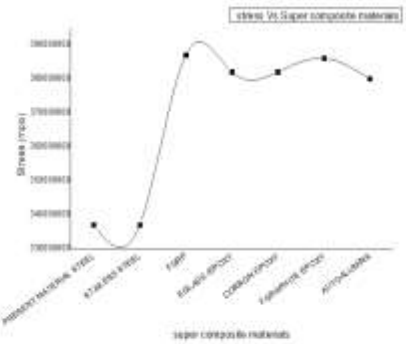

figure 10.2 Inner spring stress VS. super composite

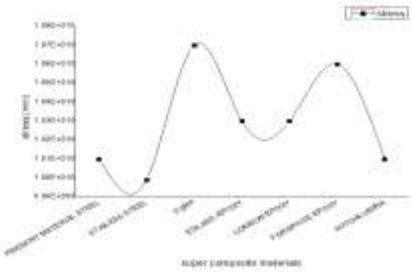

figure 10.3 outer spring stress Vs super composite

Total deformation Vs super composite materials

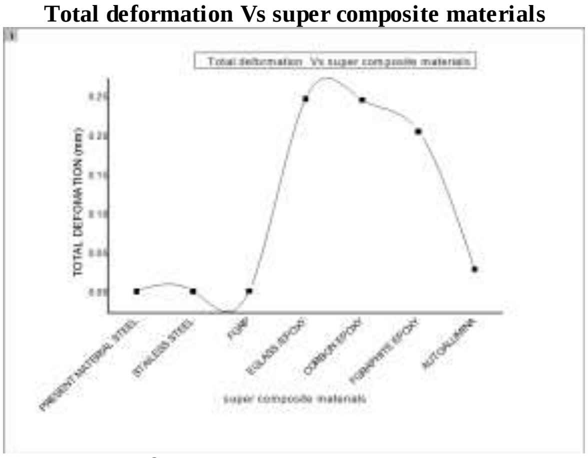

Fig 10.4 flat spring stress Vs super composite

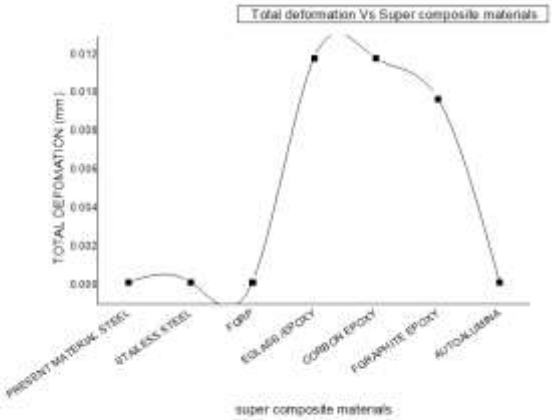

figure 10.5 Inner spring stress Vs. super composite

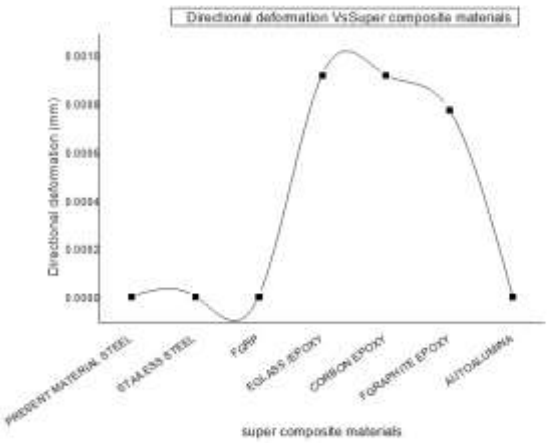

Fig 1.5 Outer spring stress VS super composite

Directional deformation Vs super composite materials

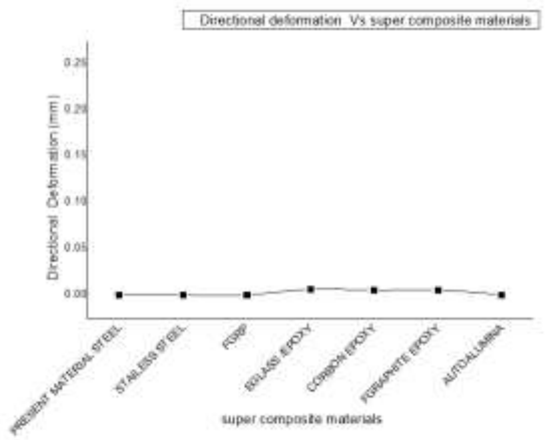

Fig 10.6 Flat spring stress vs. super composite

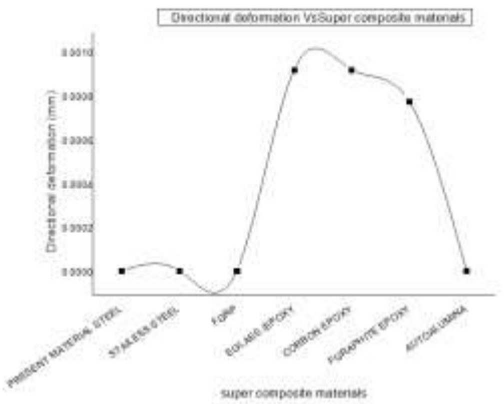

Fig:10.7 Inner spring stress's. super composite

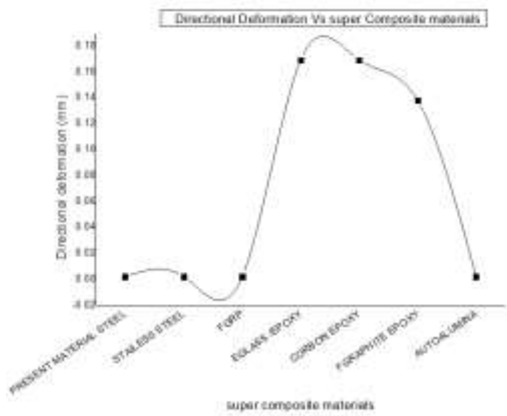

Fig 10.8 Outer spring stress VS super composite

Research Summery

Overall the project was successful because the reason for the failure was solved in the general outline. The project also produced the design of new springs to the RBO. The original project plan included the force and acceleration sensor measurements which had to be left outside the project. This was due to problematic measurement conditions which prevented the proper installation of force sensors to the spring structure. Allocated resources were also a problem for acquiring instruments. The pre-analysis with the FEM indicated that strain gage measurements could give effective information about the cause of the fracture. The project did not include fracture mechanics approach, because there were no broken springs available. Fracture mechanics could have also verified results from measurements and calculations. The definition of the project was carried out successfully even though it changed a little during the project. Changes had no remarkable effect on the schedule. The work load stayed reasonable.

The subject of the project was quite interesting. It included new theories which were not familiar from earlier studies. This was also quite motivating because of the change to learn new and essential theories in machine design. The subject turned out to be more complex than expected. The definitions and synchronisation of theoretical and experimental conditions were probably the most challenging task. Assumptions and precise determination of the physical problem at the beginning of the project helped to stay in the right direction during the project.

REFERENCES

1. Abdullah, S. Ahmand, F.N. Jalar, A. & Chua, L.B. P2009, Life Assessment of a Para-bolic Spring Under Cyclic Strain Loading. European Journal of Scientific Research, Eurojournal Publishing. Available on: http://www.sciencedirect.com/

2. Airila, M. Ekman, K. Hautala, P. Kivioja, S. Kleimola, M. Martikka, H. Miettinen, J. Niemi, E. Ranta, A. Rinkinen, J. Salonen, P. Verho, A. Vilenius, M. & Välimaa, V P.1997, Koneenosien suunnittelu. Helsinki: Werner Söderström Osakeyhtiö.

3.Aruduru, S. P. 2004. Fatigue life calculation by rainflow cycle counting method, Mid-dle East Technical University.

4.Hoffmann, K. P.1989. An Introduction to Measurements Using Strain Gages. Darmstad: Hottier Baldwin Messtechinik gmbh.

5.Korkealaakso, P. P.2009. Real-time simulation of mobile and industrial machines using the multibody approach. Lappeenranta: Lappeenranta University of Technol-ogy.

6.Kulmala, J. 2005. Rugged design goes underground. Machinedesign.com. Read 19.2.2011. Available on: www. Machinedesign.com

7.macdonald, B P.2007. Practical Stress Analysis with Finite Elements. Ireland: Glas-nevi Publishing.

8.Nakhaie Jazar, G. P.2008 Vehicle dynamics: Theory and applications. USA: Sprin-ger. Available on: http://books.google.com

9.Outinen, H. Salmi, T. & Vulli, P. P. 2007. Lujuusopinperusteet. Tampere: Klin-gendahl Paino Oy.

10.Rouvinen, A. P.2003 Mekasnimien dynamiikan simuloinnissa sovellettavia nu-meerisia- ja mallinnusmenetelmiä. Lappeenranta: Lappeenrannan teknillinen yliopis-to, Konetekniikan osasto.

11.SFS-EN 10089 2003. Hot-rolled steels for quenched and tempered springs. Technical delivery conditions. 2007. Helsinki. Finnish standards association SFS.

12 SFS-EN ISO 26909 2010. Springs. Vocabulary. 2010 Helsinki. Finnish standards association SFS. Stress-strain-diagram, Read 15.3.2011. Available on: http://www.mathalino.com

13.Vishay Micro-Measurement. The three-wire quarter bridge circuit, Application note TT-612, 2005 Read 8.4.2011. Available on:

