Fluid Mechanics: Open Access
ISSN: 2476-2296
Kumar and Abdul Kalam, Fluid Mech Open Acc 2016, 3:1
Thesis
Open Access

Design, Analysis and Comparison between the Conventional Materials with Composite Material of the Leaf Springs

Dasari Ashok Kumar[1]* and Abdul Kalam SD[2]

[1]P.V.P Siddhartha Institute of Technology, Affiliated to JNTU, Kakinada, Vijayawada, India
[2]Department of Mechanical Engineering, P.V.P Siddhartha Institute of Technology, Affiliated to JNTU, Kakinada, and Vijayawada, India

Abstract

The automobile industry has shown increased interest in the replacement of steel springs with fiberglass composites leaf spring due to high strength to weight ratio. The main aim of the project is to find the effects of replacement of the leaf spring and composite leaf spring made of E-Glass Epoxy is carried out. Comparing the load carrying capacity, stresses, stiffness, contact stiffness, and weight savings of the composite leaf spring with that of steel leaf spring is performed. The design constraints are stresses and deflections.

Finite element analysis with 3-D model of 9.525 mm thick leaf springs by introducing the contact pair in between the leafs a non-linear static analysis for the steel and composite material was done using ANSYS. The results are compared with the theoretical values and found in permissible limit.

The analysis is performed in three phases. They are by varying the load applied on the leaf springs, by varying the normal penalty stiffness (FKN) of contact pair, by varying the thickness of the composite leaf spring.

By varying the load and normal penalty stiffness the behavior of the steel and composite material multi-leaf spring is analyzed and the results are compared and holds good for composite material.

Since the composite leaf spring at 9.525 mm thickness is having the higher stiffness than required value for the comfort ride, so it is modified by reducing the thickness of the leaf spring from 9.525 mm to 8 mm and analysis is carried out and compared. From this analysis it is found that the composite leaf spring had 29.981% lesser stresses, 12.951% of higher stiffness than that of the steel leaf spring. The obtained results for varying thickness of composite leaf spring compared to the steel leaf spring are satisfactory. Due to the thickness variation the weight reduced of 69.48% was achieved. It is found that the obtained natural frequency of 8 mm thick composite leaf spring is away from the road irregularity usually have maximum frequency (12Hz) therefore resonance will not occurs and it provides improved ride comfort.

Keywords: Analysis; Design; Conventional materials; Composite material; Stresses; Leaf springs; Multi-body-systems

Introduction

Springs are unlike other machine/structure components in that they undergo significant deformation when loaded - their compliance enables them to store readily recoverable mechanical energy. In a vehicle suspension, when the wheel meets an obstacle, the springing allows movement of the wheel over the obstacle and thereafter returns the wheel to its normal position. Springs are common also in force- displacement transducers, e.g. in weighing scales, where an easily discerned displacement is a measure of a change in force. The simplest spring is the tension bar. This is an efficient energy store since all its elements are stressed identically, but its deformation is small if it is made of metal. Unlike the constant cross- section beam, the leaf spring is stressed almost constantly along its length because the linear increase of bending moment from either simple support is matched by the beam's widening - not by its deepening, as longitudinal (Figure 1).

Leaf springs are essential elements in the suspension systems of vehicles including sport utility vehicles, trucks, and railroad vehicles. Accurate modeling of the leaf springs is necessary in evaluating ride comfort, braking performance, vibration characteristics, and stability. Though simple in appearance, a leaf spring suspension causes many problems in modeling. For dynamic simulation the vehicles are usually modeled by multi-body-systems (MBS). Most wheel/axle suspension systems can be modeled by typical multi-body-systems elements like rigid bodies, links, joints and force elements. Poor leaf spring models approximate guidance and suspension properties of the leaf spring by rigid links and separate force elements. For realistic ride and handling Simulations the deformation of the leaf springs must be taken into account.

In order to conserve natural resources and economize energy, weight reduction has been the main focus of automobile manufacturer in the present scenario. Weight reduction can be achieved primarily by the introduction of better material, design optimization and better manufacturing processes. The suspension leaf spring is one of the potential items for weight reduction in automobile as it accounts for ten to twenty percent of the unsparing weight. This helps in achieving the vehicle with improved riding qualities. It is well known that springs, are designed to absorb and store energy and then release it. Hence, the strain energy of the material becomes a major factor in designing the springs. The relationship of the specific strain energy can be expressed as:

$$U = \sigma^2 / \rho E$$

***Corresponding author:** Dasari AK, P.V.P Siddhartha Institute of Technology, Affiliated to JNTU, Kakinada, Vijayawada-520 007, India, Tel: 5712300489; E-mail: mgsateja@gmail.com

Received June 30, 2016; **Accepted** July 10, 2016; **Published** July 18, 2016

Citation: Kumar DA, Abdul Kalam SD (2016) Design, Analysis and Comparison between the Conventional Materials with Composite Material of the Leaf Springs. Fluid Mech Open Acc 3: 127.

Citation: Kumar DA, Abdul Kalam SD (2016) Design, Analysis and Comparison between the Conventional Materials with Composite Material of the Leaf Springs. Fluid Mech Open Acc 3: 127.

Where 'σ' is the strength, 'ρ' the density and 'E' the Young's modulus of the spring material. It can be easily observed that material having lower modulus and density will have a greater specific strain energy capacity. The introduction of composite materials was made it possible to reduce the weight of the leaf spring without any reduction on load carrying capacity and stiffness. Since the composite materials have more elastic strain energy storage capacity and high strength-to-weight ratio as compared to those of steel.

Composite materials

Composite materials (or composites for short) are engineered materials made from two or more constituent materials with significantly different physical or chemical properties which remain separate and distinct on a macroscopic level within the finished structure. It is any substance employed in making some useful thing or artifact. The metals and ceramics are materials used in industries as good conductors and refractory materials respectively. Generally, a composite material is composed of reinforcement (fibers, particles, flakes, and/or fillers) embedded in a matrix (polymers, metals, or ceramics). The matrix holds the reinforcement to form the desired shape while the reinforcement improves the overall mechanical properties of the matrix. When designed properly, the new combined material exhibits better strength than would each individual material.

Definition: Fiber-reinforced composite materials consist of 'fibers' of high strength and modulus embedded in or bonded to a 'matrix' with di stinct interface (boundary) between them. In this form, both fibers and matrix retain their physical and chemical identities, yet they produce a combination of properties that cannot be achieved with either of the constituents acting alone. In general, fibers are the principal load carrying members, while the surrounding matrix keeps them in the desired location and orientation, acts as a load transfer medium between them and protects them from environmental damages due to elevated temperatures and humidity, for example. Thus even though the fibers provide reinforcement for the matrix, the later also serves a number of useful functions in a fiber-reinforced composite material (Figure 2).

The fibers can be incorporated into a matrix either in continuous lengths or discontinuous lengths. The principal fibers in commercial use are various types of glass, carbon and Kevlar fibers. Other fibers such as boron, silicon carbide, aluminum oxide are used in limited quantities. The matrix material may be a polymer, a metal, or a ceramic.

Common categories of composite materials

Based on the form of reinforcement, common composite materials can be classified as follows (Figure 3):

- Fibers as the reinforcement (Fibrous Composites):
    - Random fiber (short fiber) reinforced composites.
    - Continuous fiber (long fiber) reinforced composites.
- Particles as the reinforcement (Particulate composites).
- Flat flakes as the reinforcement (Flake composites).
- Fillers as the reinforcement (Filler composites).

Fiber reinforced composites

Fiber reinforced polymers (FRP) are composite materials with a polymer matrix and a glass, carbon or aramid fiber reinforcement. Common uses for FRPs generally occur in the aerospace, automotive and marine industries as low weight, high strength materials. The durability is a function of both the matrix and the fiber making them much more durable than the fibers on their own. The strength, however, is more influenced by the fibers making them very strong in tension.

FRPs are used in civil infrastructure for reinforcement for concrete patching, cables on bridges, and complete bridges. The major advantages to FRPs over steel are that the material can be more specifically tailored to the loads for the system, a resistance to corrosion, an increase in material lifetime and durability, and a decrease in construction time and cost. Unfortunately very little long term testing has been performed to ascertain the aging characteristics and limitations of the materials. Additionally, FRP's short term and long term environmental aging is still not completely understood.

The essence of fiber-reinforced composite technology is the ability to put strong stiff fibers in the right place in the right orientation and right volume fraction.

Materials

Major constituents in fiber-reinforced composite materials are the reinforcing fibers and a matrix, which acts as a binder for the fibers. Other constituents that may be found are coupling agents, coatings,

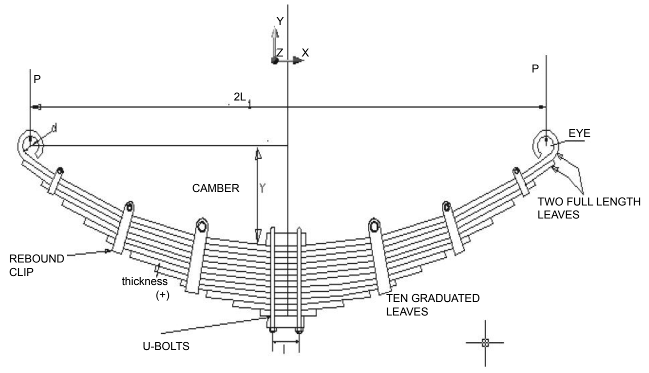

Figure 1: Leaf spring model.

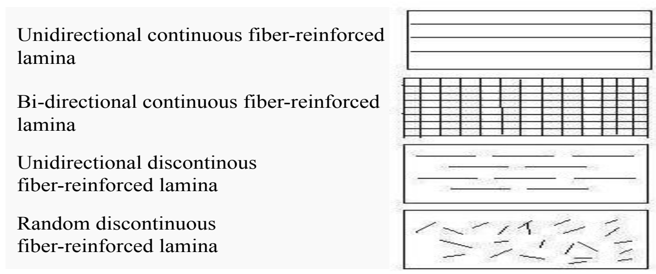

Figure 2: Types of fiber-reinforcement.

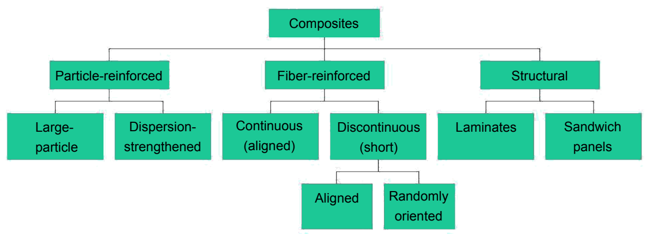

Figure 3: Classification of composite material.

and fillers. Coupling agents and coatings are applied on the fibers to improve their wetting with the matrix as well as to promote bonding across the fiber/matrix interface. This in turn promotes a better load transfer between the fibers and the matrix. Fillers are used primarily to reduce cost and improve the dimensional stability.

Fibers

Fibers are the principal constituents in a fiber-reinforced composite material. They share a major portion of the load which is acting on a composite structure. Proper selection of the type, amount, and orientation of fibers is very important since it influences the following characteristics of a composite laminate: specific gravity, tensile strength and modulus, compressive strength and modulus, fatigue strength, as well as fatigue failure mechanisms, electrical and thermal conductivities, and cost.

Glass fibers

Glass fibers are the most common of all reinforcing fibers for polymeric (plastic) matrix composites (PMC). The principle advantages of glass fibers are low cost, high tensile strength, high chemical resistance, and excellent insulating properties. The disadvantages are low tensile modulus, relatively high specific gravity (among the commercial(fibers), sensitivity to abrasion with handling which frequently decreases its tensile strength, relatively low fatigue resistance and high hardness, which causes excessive wear on moulding dies and cutting tools. The two types of glass fibers commonly used in the fiber-reinforced plastics (FRP) industry are E Glass and S Glass. Another type known as C glass is used in chemical applications requiring greater corrosion resistance to acids than is provided by E glass. E glass has the lowest cost of all commercially available reinforcing fibers, which is the reason for its widespread use in fiber-reinforced polymer industry. S glass, originally developed for aircraft components and missile castings, has the largest tensile strength among all fibers in use. A lower cost version of S Glass, called S-2 Glass, has been made available in recent years. Although S-2 Glass is manufactured with less stringent non-military specifications, its tensile strength and modulus are similar to those of S Glass. In the present work for the analysis of leaf spring E-Glass/Epoxy composite material is taken.

Advantages and disadvantage of composites

Advantages: Summary of the advantages exhibited by composite materials, which are of significant use in aerospace industry are as follows:

- High resistance to fatigue and corrosion degradation.
- High 'strength or stiffness to weight' ratio. As en umerated above, weight savings are significant ranging from 25-45% of the weight of conventional metallic designs.
- Due to greater reliability, there are fewer inspections and structural repairs.
- Directional tailoring capabilities to meet the design requirements. The fibre pattern can be laid in a manner that will tailor the structure to efficiently sustain the applied loads.
- Fibre to fibre redundant load path.
- Improved dent resistance is normally achieved. Composite panels do not sustain damage as easily as thin gage sheet metals.
- Thermoplastics have rapid process cycles, making them attractive for high volume commercial applications that traditionally have been the domain of sheet metals. Moreover, thermoplastics can also be reformed.
- Composites are dimensionally stable i.e., they have low thermal conductivity and low coefficient of thermal expansion. Composite materials can be tailored to comply with a broad range of thermal expansion design requirements and to minimize thermal stresses.
- The improved weather ability of composites in a marine environment as well as their corrosion resistance and durability reduce the down time for maintenance.
- Close tolerances can be achieved without machining.
- Improved friction and wear properties.
- The above advantages translate not only into airplane, but also into common implements and equipment such as a graphite racquet that has inherent damping, and causes less fatigue and pain to the user.

Disadvantage: Some of the associated disadvantages of advanced composites are as follows:

· High cost of raw materials and fabrication.

· Composites are more brittle than wrought metals and thus are more easily damaged.

· Transverse properties may be weak.

· Matrix is weak, therefore, low toughness.

· Reuse and disposal may be difficult.

· Repair introduces new problems, for the following reasons:

· Materials require refrigerated transport and storage and have limited shelf life.

· Hot curing is necessary in many cases requiring special tooling.

· Hot or cold curing takes time.

· Analysis is difficult.

· Matrix is subject to environmental degradation.

Selection of composite material

Based on the advantages discussed earlier, there are a number of composite materials having the required specifications. Some of the materials like E-Glass/Epoxy, graphite epoxy, carbon epoxy etc. among these E-Glass epoxy composite materials are taken for designing the leaf spring. The table shows the properties of the E-glass/Epoxy material used for the design of composite leaf spring (Table 1).

Contact overview

Contact problems are highly nonlinear and require significant computer resources to solve. It is important to understand the physics of the problem and take the time to set up the model to run as efficiently as possible. Contact problems present two significant difficulties. First, we generally do not know the regions of contact until we run the problem. Depending on the loads, material, boundary conditions, and other factors, surfaces can come into and go out of contact with each other in a largely unpredictable and abrupt manner. Second, most contact problems need to account for friction. There are several friction laws and models to choose from, and all are nonlinear. Frictional response can be chaotic, making solution convergence difficult.

Sl No	Properties	Value
1	Tensile modulus along X-direction (Ex).Mpa	34000
2	Tensile modulus along Y-direction (Ex).Mpa	6530
3	Tensile modulus along Z-direction (Ex).Mpa	6530
4	Tensile strength of the material. MPa	900
5	Compressive strength of the material. MPa	450
6	Shear modulus along XY-direction (Gxy),MPa	2433
7	Shear modulus along YZ-direction (Gxy),MPa	1698
8	Shear modulus along ZX-direction (Gxy),MPa	2433
9	Poisson ratio along XY-direction (Nuxy)	0.217
10	Poisson ratio along YZ-direction (Nuxy)	0.366
11	Poisson ratio along ZX-direction (Nuxy)	0.217
12	Mass density of the material (ρ). Kg/mm^3	$2.6 \cdot 1^{0-}6$
13	Flexural modulus of the material. MPa	40000
14	Flexural Strength of the material. MPa	1200

Table 1: Properties of E-Glass/Epoxy.

General contact classification: Contact problems fall into two general classes: rigid-to-flexible and flexible-to-flexible. In rigid-to-flexible contact problems, one or more of the contacting surfaces are treated as rigid (i.e., it has a much higher stiffness relative to the deformable body it contacts). In general, any time a soft material comes in contact with a hard material, the problem may be assumed to be rigid-to-flexible. Many metal forming problems fall into this category. The other class, flexible-to-flexible, is the more common type. In this case, both (or all) contacting bodies are deformable (i.e., have similar stiffness). An example of a flexible-to-flexible contact is bolted flanges (Figure 4).

Contact pair: In studying the contact between two bodies, the surface of one body is conventionally taken as a contact surface and the surface of the other body as a target surface. For rigid-flexible contact, the contact surface is associated with the deformable body; and the target surface must be the rigid surface. For flexible-flexible contact, both contact and target surfaces are associated with deformable bodies. The contact and target surfaces constitute a "Contact Pair". The CONTA169 contact element is associated with the 3-D target segment elements (TARGE174) using a shared real constant set number. This element is located on the surface of 3-D solid. It has the same geometric characteristics as the underlying elements.

Principles of simulating contact between parts: The ANSYS finite element analysis (FEA) program offers a variety of elements designed to treat cases of changing mechanical contact between the parts of an assembly or between different faces of a single part. FEA analysts are frequently faced with modeling situations where changing contact cannot be assumed negligible and ignored. Finding the best choices for contact elements, element options, solver, and solution options can drastically improve the model's performance and reduce the analyst's frustration with a contact simulation model. Contact elements can be grouped into four general categories based on increasing levels of Sophistication or complexity:

"Surface to Surface": This option indicates the contact occurs when the nodes on the secondary part try to pass through the surface of the primary part, and vice versa. The nodes on the primary part can contact the surface of the secondary part.

"Point to Surface": This option indicates the contact occurs when nodes on the secondary part try to pass through the surface of the primary part. The nodes on the primary part/surface can pass through the secondary surface.

"Surface to Surface (enhanced)": This variation of surface to surface contact may prevent penetration.

"Point to Surface (enhanced)": This variation of point to surface contact may prevent penetration.

"Point to Point": This method is best suited for situations in which the two surfaces experience negligible sliding relative to each other. This option indicates that the contact occurs when the nodes on the secondary part try to pass through the nodes on the primary part.

The order of these categories also reflects the history of contact element development over the past thirty years: from the initial cases developed for treatment of contact based on node point separation, to the most recent, general behavior elements which consider lower-order or higher order, quadrilateral or triangular surfaces with contact checking occurring at the face integration points.

Surface-to-surface contact: In order for loads to be transferred between elements, the nodes must be connected together. For example, if two bodies begin an analysis separated, no interaction will occur during the analysis. Surface-to-surface contact in Mechanical Event Simulation and nonlinear stress analysis (but not natural frequency analysis) allows you to create pairs of surfaces that may come into contact with each other during the analysis thereby connecting the nodes on the surfaces together. The processor will determine the distance between the nodes on this surface at each time step of the analysis. When the nodes are sufficiently close to each other, a force will be applied to prevent penetration. Before the user starts a contact analysis, he or she must clearly identify where the contact interaction might occur during the analysis.

Proper mesh: In order to perform an accurate contact analysis, a reasonably smooth contact surface and a uniform mesh over the contact surfaces are highly recommended. Furthermore, since surface mesh alignment can impact the performance of the whole analysis, surface meshing on each adjacent contact surface should be carefully done to achieve a better convergence rate. In linear contact, the mesh on the two surfaces should be perfectly matched to each other as shown in Figure 5 (a). However, in Mechanical Event Simulation a perfectly matched mesh could cause severe contact chattering. When this chattering occurs, it inevitably results in a very poor convergence rate (i.e., longer runtime). In order to avoid or at least minimize this problem, when small relative motion between the contact surfaces is expected, it is recommended that the mesh should have the nodes on one surface located in the middle of the nodes on the other surface as shown in Figure 5 (b).

In this problem involving contact between two boundaries, one of the boundaries is conventionally established as the "target" surface, and the other as the "contact" surface. These two surfaces together comprise

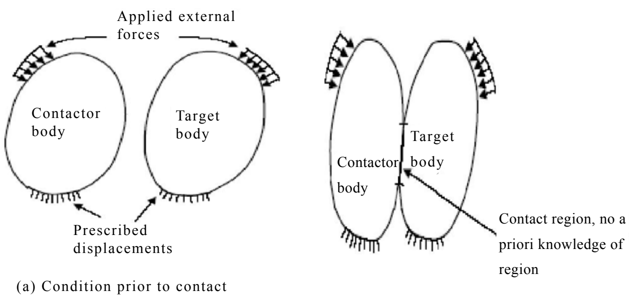

Figure 4: Contact pair formulation.

the "contact pair". Use TARGE169 with CONTA174 to define a 3-D contact pair (Figure 6).

CONTA174-3d 8-node surface-to-surface contact: CONTA174 is an 8-node element that is intended for general rigid-flexible and flexible-flexible contact analysis. In a general contact analysis, the area of contact between two (or more) bodies is generally not known in advance. CONTA174 is applicable to 3-D geometries. It may be applied for contact between solid bodies or contact bodies.

Element description: CONTA174 is used to represent contact and sliding between 3-D "target" surfaces (TARGE174) and a deformable surface, defined by this element. The element is applicable to 3-D structural and coupled field contact analyses. This element is located on the surfaces of 3-D solid or shell elements with midside nodes. It has the same geometric characteristics as the solid or shell element face with which it is connected. Contact occurs when the element surface penetrates one of the target segment elements (TARGE170) on a specified target surface. Coulomb and shear stress friction is allowed (Figure 7).

CONTA174 assumptions and restrictions:

- The 3-D contact element must coincide with the external surface of the underlying solid or shell element.
- This element is nonlinear and requires a full Newton iterative solution, regardless of whether large or small deflections are specified.
- The normal contact stiffness factor (FKN) must not be so large as to cause numerical instability.
- FTOLN(Penetration tolerance factor), PINB (Pinball region),and FKOP(Contact opening stiffness)can be changed between load steps or during restart stages.
- The value of FKN can be smaller when combined with the Lagrangian multiplier method, for which TOLN must be used.
- You can use this element in nonlinear static or nonlinear full transient analyses.
- In addition, you can use it in modal analyses, eigenvalue buckling analyses, and harmonic analyses. For these analysis types, the program assumes that the initial status of the element (i.e., the status at the completion of the static prestress analysis, if any) does not change.

This element allows birth and death and will follow the birth and death status of the underlying solid, shell, beam or target elements.

TARGE170 - 3-d target segment: In studying the contact between two bodies, the surface of one body is conventionally taken as a contact surface and the surface of the other body as a target surface. The "contact-target" pair concept has been widely used in finite element simulations. For rigid-flexible contact, the contact surface is associated with the deformable body; and the target surface must be the rigid surface. For flexible-flexible contact, both contact and target surfaces are associated with deformable bodies. The contact and target surfaces constitute a "Contact Pair".

Targe170 element description

TARGE170 is used to represent various 3-D "target" surfaces for the associated contact elements. The contact elements themselves overlay the solid elements describing the boundary of a deformable body and are potentially in contact with the target surface, defined by TARGE170. This target surface is discredited by a set of target segment elements (TARGE170) and is paired with its associated contact surface via a shared real constant set. It can be used in any translational or rotational displacement, temperature, voltage, and magnetic potential on the target segment element. For rigid target surfaces, these elements can easily model complex target shapes. For flexible targets, these elements will overlay the solid elements describing the boundary of the deformable target body (Figure 8).

Targe170 assumptions and restrictions

- Should not change real constants R1 or R2, either between load steps or during restart stages, otherwise ANSYS assumes the radii of the primitive segments varies between the load steps.

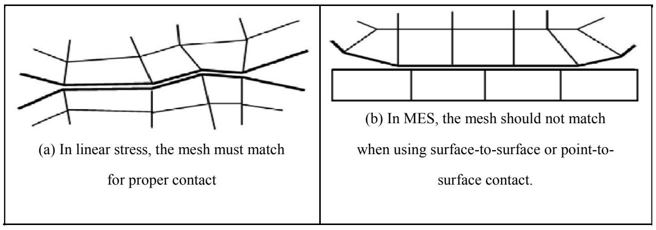

Figure 5: Proper mesh for contact pairs.

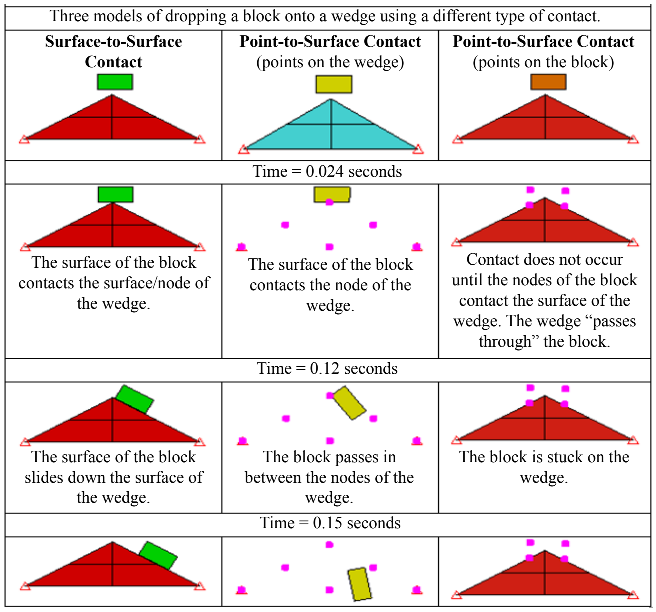

Figure 6: Comparison of contact types.

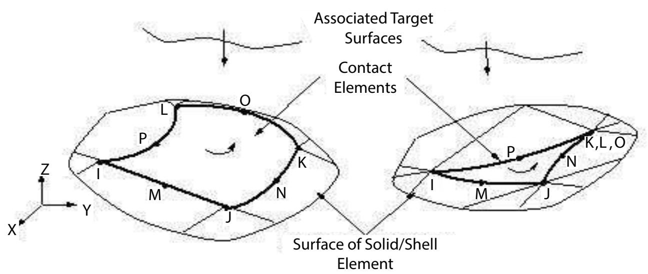

Figure 7: CONTA174 geometry.

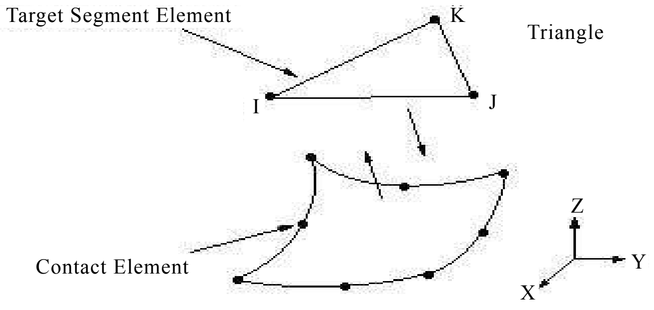

Figure 8: TARGE170 geometry.

When using direct generation, the real constants for cylinders, cones, and spheres may be defined before the input of the element nodes. If multiple rigid primitives are defined, each having different radii, they must be defined by different target surfaces.

No external forces can be applied on target nodes except on a pilot node. To ensure the correct behavior, apply all boundary conditions to a pilot node.

Literature Survey

Introduction

Aim of this chapter is to discuss about the literatures published on the design, analysis of leaf spring by different investigators earlier and to know the scope of future works. A good number of publications are available on this topic which indicates the importance and necessary to analysis the leaf springs which are used for suspension system an automobile industry.

Discussions

BD yang et al. [1] proposed that Mechanical systems in which moving components are mutually constrained through contacts often lead to complex contact kinematics involving tangential and normal relative motions. A friction contact model is proposed to characterize this type of contact kinematics. The stick-slip friction phenomenon is analyzed by establishing analytical criteria that predict the transition between stick- slip and separation of the interface.

MS Kumar and S Vijayarangan [2] describes static and fatigue analysis of steel leaf spring and composite multi leaf spring made up of glass fibre reinforced polymer using life data analysis Static analysis of 2-D model of conventional leaf spring is performed using ANSYS 7.1 and compared with experimental results. And they fabricated a composite multi leaf spring using E-glass/Epoxy unidirectional laminates. The load carrying capacity, stiffness and weight of composite leaf spring are compared with that of steel leaf spring analytically and experimentally. Finite element analysis with full bump load on 3-D model of composite multi leaf spring is done and the analytical results are compared with experimental results. Fatigue life of steel leaf spring and composite leaf is also predicted. A weight reduction of 68.15% is also achieved by using composite leaf spring. It is also concluded that fatigue life of composite is more than that of conventional steel leaf spring.

Ralf Diekmann et al. [3] proposed an efficient and accurate contact algorithm is essential for FE-simulation several moving bodies or fragmentation processes of brittle material. In this paper, a brief description of a contact formulation used within an explicit, dynamic 2D FE-code is given. The contact algorithm is a modified Lagrangian method; it ensures non-penetration after each step of the explicit time integration. It combines straight-line and C1-continuous (smoothed) boundary approximations. A flexible dynamic data structure allows dynamically growing surfaces which are needed to simulate fragmentation processes. A new global contact search algorithm based on position codes is investigated.

Peiyong Qin et al. [4] said that leaf spring design was mainly based on simplified equations in which the models were limited to the three-link mechanism assumption and linear beam theory. In this the detailed finite element modeling and analysis of a two-stage multi-leaf spring, a leaf spring assembly, and a Hotchkiss suspension using ABAQUS are presented. Non-linearity from large deformation, interleaf contact, and friction ware explained. Stresses and strains under different loads were analyzed. The simulation method presented can be used for the development of leaf springs and Hotchkiss suspensions. Suspension characteristics can be verified after the leaf geometry is designed. In addition, the rates predicted can be used in full vehicle NVH models or multi-body dynamic models. Over all, the method introduced can help to reduce product development time and costs significantly.

Hui-Lin Xing and Akitake Makinouchi [5] based on the characters of the explicit time integration algorithm, a reliable and efficient contact element strategy, named as node-to-point contact element strategy, is proposed and applied to handle the static or quasi-static multi deformation-body contact with friction. The Coulomb friction model governs the friction behavior with an additional limit on the allowable shear stress, which is treated as a flow plasticity rule. The penalty method is adopted to impose the normal and the sticking contact. Finally, numerical examples of contact between finite deformation bodies are presented to show the efficiency and stability of this algorithm.

Georg Rill et al. [6] develops "Leaf Spring Modeling for Real Time Applications". Even though it is the oldest type of automotive suspension; leaf springs continue to be a popular choice for solid axles. Though simple in appearance, a leaf spring suspension causes many problems in modeling. For dynamic simulation the vehicles are usually modeled by multi-body-systems (MBS). Most wheel/axle suspension systems can be modeled by typical multi-body-systems elements like rigid bodies, links, joints and force elements. Poor leaf spring models approximate guidance and suspension properties of the leaf spring by rigid links and separate force elements. For realistic ride and handling simulations the deformation of the leaf springs must be taken into account.

GS Shankar and S Vijayarangan [7] presented a low cost fabrication of complete mono composite leaf spring with bonded end joints, and general study on the analysis and design. A single leaf with variable thickness and width for constant cross sectional area of unidirectional glass fiber reinforced plastic (GFRP) with similar mechanical and geometrical properties to the multi-leaf spring, was designed, fabricated (hand-lay-up technique) and tested. The results showed that an spring width decreases hyperbolically and thickness increases linearly from the spring eyes towards the axle seat. They demonstrated that composites can be used for leaf springs for light weight vehicles and meet the requirements, together with substantial weight savings. And they observed that the composite leaf spring is lighter and more economical than the conventional steel spring with similar design specifications.

PC Pandey [8], in his class the General introduction and concept of composites, and Concept of Composite materials, material properties that can be improved by forming a composite material and its engineering potential has been studied. And different types of composites materials are explained. The different classification based on matrix material and reinforcements are described briefly. In this a brief explanation about the fibre reinforced Polymer (FRP) Laminated composites and their behaviours. A brief explanation about the design concepts and manufacturing processes for the composite materials. And finally the testing procedures of the composite materials and engineering application are clearly explained.

MA Rahman et al. [9] gave a brief explanation on the response of a leaf spring of parabolic shape which is made of highly elastic steel. By using the numerical simulation of both small and large deflection theories to calculate the stresses and deflection of the beam. The actual bending stresses at fixed end are lesser than the traditional leaf spring having the same volume of material. The effects of two vitally important factors, namely, the end-shortening and geometric nonlinearity, on the response of parabolic shaped variable cross section, have been demonstrated by numerical analysis. Nonlinear solution plays vital role in determining the true stresses in highly flexible structures. It is found that the response, in terms of stress and deflection, of the proposed parabolic leaf spring is not significantly changed from that of a traditional leaf spring. Therefore, it justifies the use of such a parabolic contour, especially, in terms of economy and light weight of the leaf spring.

M Hosain [10] explaining the effects of replacement of the steel leaf spring with a composite one, with the same vertical stiffness, on the different aspects of the vehicle dynamics are discussed. For this purpose, the discreet flex models of the steel and composite leaf springs, the full vehicle model and appropriate road models are constructed with ADAMS solution. Dynamic analysis such as; handling, ride and lateral stability are simulated by ADAMS based on standard road tests. The results show that the composite leaf spring is better as the durability point of view. However, its roll behavior is worse in normal speed of 80 Km/h and this will weaken the vehicle handling.

ML Aggarwala and PS Chawlab [11] explained the Shot peening has used as a low cost and simple method for increasing fatigue strength of the leaf springs. During a survey, it has been found that fretting fatigue failure of semi-elliptical leaf springs in commercial vehicles occur between 3-6 years. The bending fatigue strength of EN45A spring steel parabolic leaf springs is found to be much higher as compared to semi-elliptical leaf springs. In this work they discusses the elimination of fretting fatigue between leaves of EN45A spring steel shot peened leaf springs by using taper leaves and rubber pads. From this they concluded that Fatigue life of semi-elliptical leaf springs is found to be lower due to fretting fatigue between mating leaves, and parabolic leaf springs possess higher damping factors than semi- elliptical leaf springs. And Fatigue life of tapered leaves in parabolic leaf springs is found to be more than straight leaves of semi-elliptical leaf springs due to higher damping factor and absence of fretting fatigue.

Conclusions and present work

From the review of the literature presented in the previous section, it has been observed that the analysis of the leaf springs by using finite element analysis technique and developing computer program for finding out the response for static and dynamic analysis of the leaf spring. Here more work has been done on the static analysis to maintain the design in safe condition. The formulation of finite element technique was used to developed the contact pairs between the leafs to analysis the behavior of the leaf spring [1-7]. The static analysis, model analysis and the life prediction, fatigue life prediction of the leaf springs was presented [8-10]. Similarly from all other papers presented it is found there is a lot of scope for analysis of leaf springs of steel, composite materials based on their load variation, contact stiffness variation, thickness of leaf variation, leaf width variation, leaf length variation.

Suspension Leaf Spring

Suspension leaf spring

The leaf spring main purpose is to filter out the axle excitation before these disturbances reach the chassis. There are a variety of different suspensions used on vehicles. However, some types of suspensions have grown more popular than others. In the truck /car industry the overwhelming majority are leaf springs. Leaf springs are less expensive, simpler and more reliable than any other common suspension. In addition they act as both spring and damper simultaneously, thus, reducing or eliminating the need for independent shock absorbers.

Suspension model

All suspension systems contain two main ingredients, a spring component and a damper component. The suspension's main purpose is to filter out the axle excitation before these disturbances reach the chassis. There are a variety of different suspensions used on vehicles. However, some types of suspensions have grown more popular than others. In the truck /car industry the overwhelming majority are leaf springs. Leaf springs are less expensive, simpler and more reliable than any other common suspension. In addition they act as both spring and damper simultaneously, thus, reducing or eliminating the need for independent shock absorbers. Leaf springs are commonly used for wheel suspensions in vehicles in order to provide cushioning against uneven road surfaces. Such vehicles may include, but are not limited to, passenger vehicles, trucks, and other utility vehicles, and may also include railcars and similar vehicles.

Leaf spring suspension

a) Single axle leaf spring.

b) Tandem Leaf Spring/Short rocker.

c) Tridem Leaf Spring/Short Rocker.

Leaf spring model

A leaf is made up of laminated strips of curved steel. The chassis supports the two ends and middle of the leaf spring is connected to the axle. As the leaf spring is compressed, the steel leaves bend acting as springs, and the leaves slide across each other dissipating energy through coulomb friction. The mathematical leaf-spring model used in this study is the semi-analytic model based on the Euler beam theory (Figure 9).

Types of leaf springs

There are four basic types of leaf spring systems:

- **Multi-Leaf Spring** - This type of leaf spring has more than 1 leaf in its assembly. It consists of a center bolt that properly aligns the leaves and clips to resist its individual leaves from twisting and shifting.
- **Mono Leaf spring** - Consists of one main leaf where the material's width and thickness are constant. Example - the leaf will be 2 ½" wide throughout its length, and 0.323" in thickness throughout its entire length. The spring rate is lighter than

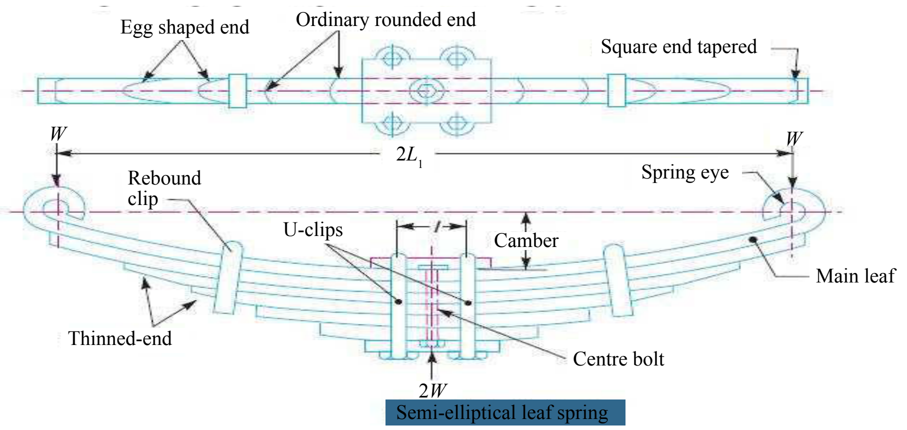

Figure 9: Semi elliptical leaf spring.

other styles of leaf springs and usually requires a device to control positive and negative torque loads as well as requiring coil springs to hold the chassis at ride height.

- **Parabolic Single leaf** - Consists of one main leaf with a tapered thickness. This style is sufficient to control axle torque and dampening, while maintaining ride height. The advantage of this style is that the spring is lighter than the multi-leaf.
- **Fiberglass Leaf Spring** - The fiberglass leaf spring is made of a mixture of plastic fibers and resin; it is lighter than all other springs. However, the cost is three times greater. In addition, fiberglass springs are sensitive to heat.

Characteristics of a good suspension include

- Maximum deflection consistent with required stability.
- Compatible with other vehicle components in terms of overall ride.
- Minimum weight.
- Low maintenance and operating costs.
- Minimize tire wear.
- Minimize wheel hop.
- Low initial cost.

Functions of leaf spring

- Support the weight of the vehicle.
- Provide adequate stability and resistance to side away and rollover.
- Resist cornering effects when negotiating a curve.
- Provide cushioning.

Leaf springs are designed to

- Connect the axle to the vehicle.
- Transfer driving and braking forces between frame and axle.
- Resist drive and brake torque, known as wrap up.
- On drive axles provide minimum changes in drive axle pinion and limit movement of drive axle slip splines.

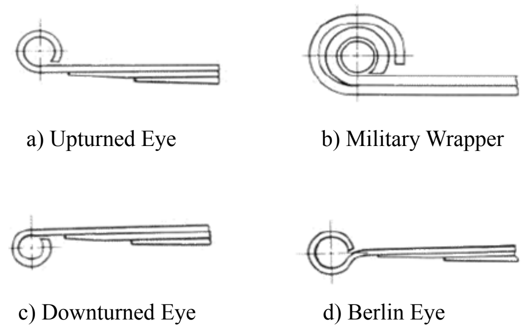

Figure 10: Types of spring eyes.

- On steering axles, they maintain the proper wheel caster and camber.

Spring eyes

A spring eye is essentially the end of a leaf spring bended into a circular shape to allow rotation about the spring eye. The main types of spring eye designs are upturned, military wrapper, down turned, and Berlin eyes (Figure 10).

Factor of safety

The designer must take into account the factor of safety when designing a structure. Since, composites are highly orthotropic and their fractures were not fully studied, factor of safety was variedly taken and analyzed here. For this problem as the factor of safety of 4 is taken in to consideration.

Introduction to Finite Element Modeling

Introduction to finite element method

The finite element method is a numerical procedure for analyzing structures and continua. Usually, the problem addressed is too complicated to be solved satisfactorily by classical analytical methods. The problem may concern stress analysis, heat conduction, or any of several other areas. The finite element procedure produces many simultaneous algebraic equations, which are generated and solved on a digital computer. Results are rarely exact. However, errors are decreased by processing more equations, and results accurate enough for engineering purposes are obtainable at reasonable cost.

The finite element method originated as a method of stress analysis. Today, finite elements are also used to analyze problems of heat transfer, fluid flow, lubrication, electric- and magnetic fields and many others. Problems that previously were utterly intractable are now solved routinely. Finite element procedures are used in the design of buildings, electric motors, heat engines, ships, airframes, and space craft's. Manufacturing companies and large design offices typically have one or more large finite element programs in-house. Smaller companies usually have access to a large program through a commercially computing center or use a smaller program on a personal computer.

The finite element method is a method of piecewise approximation in which the approximating function φ is formed by connecting simple functions, each defined over a small region (element). A finite element is a region in space in which a function φ is interpolated from nodal values of φ on the boundary of the region in such a way that inter element continuity of φ tends to be maintained in the assemblage.

A finite element analysis typically involves the following steps. Steps 1, 4, and 5 require decisions by the analyst and provide input data for the computer program. Steps 2, 3, 6, and 7 are carried automatically by the computer program.

- Divide the structure or continuum into finite elements. Mesh generation programs, called preprocessors, help the user in doing this work.
- Formulate the properties of each element. In stress analysis, this means determining nodal loads associated with all element deformation states that are allowed. In Heat transfer, it means determining nodal heat fluxes associated with all element temperature fields that are allowed.
- Assemble elements to obtain the finite element model of the structure.
- Apply the known loads: nodal forces and/or moments in stress analysis. Nodal heat fluxes in heat transfer.
- In stress analysis, specify how the structure is supported. This step involves setting several nodal displacements to know values (which are often zero). In heat transfer, where typically certain temperatures are known impose all known values of nodal temperatures.
- Solve simultaneous linear algebraic equations to determine nodal DOF (nodal displacements in stress analysis, nodal temperatures in heat transfer).
- In stress analysis, calculate element strains from the nodal DOF and element displacement field interpolation, and finally calculate stresses from strains. In heat transfer, calculate element, heat fluxes from the nodal temperatures and the element temperature field interpolation. Output interpretation programs, called postprocessors, help the user sort the output and display it in graphical form.

The power of the finite element method resides principally in its versatility. The method can be applied to various physical problems. The body analyzed can have arbitrary shape, loads, and support conditions. The mesh can mix elements of different types, shapes, and physical properties. This great versatility is contained within a single computer program. User-prepared input data controls the selection of problem type, geometry, boundary conditions, element selection, and so on.

Another attractive feature of finite elements is the close physical assemblence between the actual structure and its finite element model. The model is not simple an abstraction. This seems especially true in structural mechanics, and may account for the finite element method having its origins there.

The finite element method also has disadvantages. A specific numerical result is found for a specific problem: a finite element analysis provides no closed-form solution that permits analytical study of the effects of changing various parameters. A computer, a reliable program, and intelligent use are essential. A general -purpose program has extensive documentation, which cannot be ignored. Experience and good engineering judgment are needed in order to define a good model. Many input data are required and voluminous output must be sorted and understood.

Introduction to Ansys

ANSYS is a large-scale multipurpose finite element program developed and maintained by ANSYS Inc. to analyze a wide spectrum of problems encountered in engineering mechanics.

Program organization: The ANSYS program is organized into two basic levels:

· Begin level

· Processor (or Routine) level

The Begin level acts as a gateway into and out of the ANSYS program. It is also used for certain global program controls such as changing the job name, clearing (zeroing out) the database, and copying binary files. When you first enter the program, you are at the Begin level.

At the Processor level, several processors are available. Each processor is a set of functions that perform a specific analysis task. For example, the general preprocessor (PREP7) is where you build the model, the solution processor (SOLUTION) is where you apply loads and obtain the solution, and the general postprocessor (POST1) is where you evaluate the results of a solution. An additional postprocessor, POST26, enables you to evaluate solution results at specific points in the model as a function of time (Table 2).

Material models: ANSYS allows several different material models like:

· Linear elastic material models (isotropic, orthotropic, and anisotropic).

· Non-linear material models (hyper elastic, multilinear elastic, inelastic and viscoelastic).

· Heat transfer material models (isotropic and orthotropic).

· Temperature dependent material properties.

· Creep material models.

Loads: The word loads in ANSYS terminology includes boundary conditions and externally or internally applied forcing functions, as illustrated in Loads. Examples of loads in different disciplines are:

· Structural: displacements, forces, pressures, temperatures (for thermal strain), gravity.

· Thermal: temperatures, heat flow rates, convections, internal heat generation, infinite surface.

· Magnetic: magnetic potentials, magnetic flux, magnetic current segments, source current density, infinite surface.

Processor	Function	GUI Path	Command
PREP7	Build the model (geometry,materials, etc.)	Main Menu> Preprocessor	/PREP7
SOLUTION	Apply loads and obtain thefinite element solution	Main Menu> Solution	/SOLU
POST1	Review results over the entire model at specific time points	Main Menu> General Postproc	/POST1
POST26	Review results at specific points in the model as a function of time	Main Menu> Time Hist Postpro	/POST26
OPT	Improve an initial design	Main Menu> Design Opt	/OPT
PDS	Quantify the effect of scatter and uncertainties associated with input variables of a finite element analysis on the results of the analysis	Main Menu> Prob Design	/PDS
AUX2	Dump binary files in readable form	Utility Menu> File> List> Binary Files Utility Menu> List> Files> Binary Files	/AUX2
AUX12	Calculate radiation view factors and generate a radiation matrix for a thermal analysis	Main Menu> Radiation Matrix	/AUX12
AUX15	Translate files from a CAD or FEA program	Utility Menu> File> Import	/AUX15
RUNSTAT	Predict CPU time, wave front requirements, etc. for an analysis	Main Menu> Run-Time Stats	/RUNST

Table 2: Processors (Routines) available in ANSYS processor.

- Electric: electric potentials (voltage), electric current, electric charges, charge densities, infinite surface.
- Fluid: velocities, pressures Loads are divided into six categories: DOF constraints, forces (concentrated loads), surface loads, body loads, inertia loads, and coupled-field loads.
- A DOF constraint fixes a degree of freedom (DOF) to a known value. Examples of constraints are specified displacements and symmetry boundary conditions in a structural analysis, prescribed temperatures in a thermal analysis, and flux-parallel boundary conditions.
- A force is a concentrated load applied at a node in the model. Examples are forces and moments in a structural analysis, heat flow rates in a thermal analysis, and current segments in a magnetic field analysis.
- A surface load is a distributed load applied over a surface. Examples are pressures in a structural analysis and convections and heat fluxes in a thermal analysis.
- A body load is a volumetric or field load. Examples are temperatures and fluencies in a structural analysis, heat generation rates in a thermal analysis, and current densities in a magnetic field analysis.
- Inertia loads are those attributable to the inertia (mass matrix) of a body, such as gravitational acceleration, angular velocity, and angular acceleration. You use them mainly in a structural analysis.
- Coupled-field loads are simply a special case of one of the above loads, where results from one analysis are used as loads in another analysis. For example, you can apply magnetic forces calculated in a magnetic field analysis as force loads in a structural analysis.

Analysis types: The following types of analysis are possible using ANSYS:

- Structural Analysis: Static Analysis, Modal Analysis, Harmonic Analysis, Transient Dynamic Analysis, Spectrum Analysis, Buckling Analysis, Explicit Dynamic Analysis, Fracture mechanics, and Beam Analysis.
- Thermal Analysis: Steady-state thermal analysis, transient thermal analysis.
- CFD (Computational Fluid Dynamics) Analysis: Laminar or turbulent, Thermal or adiabatic, Free surface, Compressible or incompressible, Newtonian or Non-Newtonian, Multiple species transport.
- Several types of Electromagnetic field analysis and Coupled field analysis.

Post processing: Post processing means reviewing the results of an analysis. It is probably the most important step in the analysis, because you are trying to understand how the applied loads affect your design, how good your finite element mesh is, and so on.

Two postprocessors are available to review your results: POST1, the general postprocessor, and POST26, the time-history postprocessor. POST1 allows you to review the results over the entire model at specific load steps and sub steps (or at specific time-points or frequencies).

POST26 allows you to review the variation of a particular result item at specific points in the model with respect to time, frequency, or some other result item. In a transient magnetic analysis, for instance, you can graph the eddy current in a particular element versus time. Or, in a nonlinear structural analysis, you can graph the force at a particular node versus its deflection.

Static analysis: A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

Static analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time. The types of loading that can be applied in a static analysis include:

Externally applied forces and pressures

Steady-state inertial forces (such as gravity or rotational velocity)

Imposed (nonzero) displacements

Temperatures (for thermal strain)

Fluences (for nuclear swelling)

A static analysis can be either linear or nonlinear. All types of nonlinearities are allowed - large deformations, plasticity, creep, stress stiffening, contact (gap) elements, hyper elastic elements, and so on. This analysis gives a clear idea whether the structure or component will withstand for the applied maximum forces. If the stress values obtained in this analysis crosses the allowable value, it will result in the failure of the structure in the static condition itself. To avoid such a failure, this analysis is necessary.

Modal analysis: When an elastic system free from external forces is disturbed from its equilibrium position, it vibrates under the influence of inherent forces and is said to be in a state of free vibration. It will vibrate at its natural frequency and its amplitude will gradually become smaller with time due to energy being dissipated by motion. The main parameters of interest in free vibration are natural frequency and the amplitude. The natural frequencies and the mode shapes are important parameters in the design of a structure for dynamic loading conditions. Modal analysis is used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis.

System configuration

In the present work, the computational numerical analysis is done using ANSYS version 12.0 running on Pentium IV system, having 4GB ram and 160GB hard disk with Windows XP operating system.

Problem Definitions and Methodology

Problem definition for the leaf springs

There is currently much interest in deformation analysis of multiple elastic-plastic bodies in contact. One such case is the design and analysis of the automobile leaf springs. In order to accurately model the deformations and vibrations of the leaf springs, nonlinear finite-element procedures are need to be employed, with the advent of development of the contact analysis, it is appropriate to apply the contact analysis technique in the analysis of the leaf springs. Thus the effect of the system with contact has to be studied. Methods for modeling the contact and friction between the leaves of the spring are to be developed.

The conditions to be achieved

- Each leaf should be capable of holding both tensile and compressive loads successively.
- Stresses and deflection should be within the permissible limits.
- Perfect contact pair should be formulated.
- Leafs should always be bonded to each other.
- Converges of the model should be achieved.
- The effect of the dynamic load on the spring stiffness and the effect of the structural damping on the response of the leaf spring are to be considered.

Solution methods adopted

Problem solution approach:

In this work the standard design parameters of rear leaf spring of DODGE car manufacturers is considered. The basic understanding of the effect of leaf spring geometric variables (i.e., width, thickness, and length of each leaf, and the total number of leaves) on the stress and stiffness is clearly understood. The following steps will explain the methods followed for analysing the leaf spring.

Step 1: First the collected data is validated with the standard mathematical calculation.

Step 2: And then formulated the finite element model in the ANSYS.

Step 3: Different modeling techniques are considered for analysis are presented.

A. Initial the leaf spring is modeled and performed the analysis for different loads and compared stress and stiffness with the theoretical values.

B. Secondly the model is refined for different contact stiffness values and performed the analysis for steel material, where a parametric study is performed by keeping the contact pairs and the comparison of the stresses variation at the different stages is closely monitored.

C. The variation for the deflection, stiffness and stress at different loads and contact stiff nesses are plotted.

Step 4: The same processes was continued with varying loads, contact stiffness and thicknesses for the composite material.

Step 5: Finally it is observed that by using the composite material the leaf spring thickness can be reduced without affecting the permissible conditions.

Step 6: Finding the weight reduction obtained by using the composite material for the leaf spring.

Step 7: For maintaining the smooth riding the model analysis is carried for finding the natural frequency of both standard steel leaf spring and replaced composite material leaf spring.

Mathematical analysis

This chapter involves the determination of bending stress by using mathematical formula. Determination of length of leaf spring leaves, consequently the rotation angle and the radius of curvatures of each leaf, these are used in geometric modeling.

The Length of the leave spring leaves obtained as discussed below:

$2L_1$ = Length of span or overall length of the spring

L = Distance between centers of U-bolt. It is the ineffective length (E.L) of the leaf spring

Nf = Number of full length leaves

Ng = Number of graduated leaves

N = Total number of leaves = Nf+Ng

E.L = Effective length of the spring = $2L_1$-(2/3) L

$$\text{Length of smallest leaf} = \frac{\text{Effective Length} \times 1}{n-1} + \text{IneffectiveLength}$$

$$\text{Length of next leaf} = \frac{\text{Effective Length} \times 2}{n-1} + \text{IneffectiveLength}$$

Similarly,

$$\text{Length of (n-1)}^{\text{th}} \text{ leaf} = \frac{\text{Effective Length} \times n-1}{n-1} + \text{IneffectiveLength}$$

Length of master leaf = Length of master leaf = 2 L_1+2 Π (d + t)*2

Where,

d = Inside diameter of eye

t = Thickness of leaf

Relation between Radius of curvature (R) and chamber (Y) of the spring is given by

$$Y(2R+Y) = L_1^2$$

Where,

L_1 = Half span of spring

$$\text{Half angle} = \frac{hl \times 360}{(R \times 2\pi)}$$

Consider the standard design data. By considering the following data for our analysis (Tables 3 and 4).

Determine the length of each leaf:

$$\text{Length of smallest leaf} = \frac{\text{Effective Length} \times 1}{n-1} + \text{IneffectiveLength}$$

$$= \frac{2 \times 620.4 \ \times 1}{7} + 80$$

= 286.8 mm

$$\text{Length of } 2^{nd} \text{ leaf} = \frac{2 \times 620.4 \ \times 2}{7} + 80$$

= 493.6 mm

$$\text{Length of } 3^{rd} \text{ leaf} = \frac{2 \times 620.4 \ \times 3}{7} + 80$$

= 700.4 mm

$$\text{Length of } 4^{th} \text{leaf} = \frac{2 \times 620.4 \ \times 4}{7} + 80$$

= 907.2mm

$$\text{Length of } 5^{th} \text{leaf} = \frac{2 \times 620.4 \ \times 5}{7} + 80$$

= 1114 mm

$$\text{Length of } 6^{th} \text{leaf} = \frac{2 \times 620.4 \ \times 6}{7} + 80$$

= 1320.8mm

$$\text{Length of } 7^{th} \text{leaf} = \frac{2 \times 620.4 \ 76}{7} + 80$$

= 1320.8 mm

Relation between Radius of curvature (R) and chamber (Y) of the spring is given by

$$\text{Half angle} = \frac{hl \times 360}{(R \times 2\pi)}$$

Where,

L_1 = Half span of spring

$$\text{Half angle} = \frac{hl \times 360}{(R \times 2\pi)}$$

Dimensions of each leaf is shown in Table 5.

Problem solution approach

In this work the standard design parameters of rear leaf spring of "DODGE car" is taken. Table 1 lists the design implications that obtained from our parametric model (Table 6).

Standard materials: For the automobile, the recommended materials are: 50Cr 1, 50 cr1V/23 and 55Si 2Mn 90, all used in hardened and tempered state to derive the high strength available for these materials. The physical properties of some of these materials are given in Table 2; all the values are for oil quenched condition and for single heat only (Table 7).

From the above standard table 50Cr 1 steel is considered for leaf the spring which is having yield stress=1510 N/mm^2 taking factor of safety as 4 into consideration;

Working stress=377.5 N/mm^2

Young modules=2.1E5 N/mm^2

Density= 7.86E-6 kg/mm^2

Poison ration=0.3

Geometric Variable	Design Implications
Leaf length	2L = 1320.8 mm
Number of leaves (n)	Ng = 5
	Nf = 2
Leaf thickness (t)	T = 9.525 mm
Leaf width (w)	b = 63.5 mm

Table 3: Design implications of leaf springs geometric variables.

Material data	50cr1 steel
Young modules	2.1E5 N/mm^2
Density	7.86E-6 kg/mm^2
Poison ration	0.3
Expected load carried 2W	6000N

Table 4: 50cr1 steel material properties.

Leaf data	Full length(mm)	Half length (mm)	Half angle(Ǿ)
Full length leaf	1320.8	660.4	8.55
2nd gradient leaf	1320.8	660.4	8.53
3rd gradient leaf	1114	557	7.195
4th gradient leaf	907.2	453.6	5.748
5th gradient leaf	700.4	350.2	4.4045
6th gradient leaf	493.6	246.8	3
7th gradient leaf	286.8	142.4	1.8366

Table 5: Dimensions of each leaf.

Geometric Variable	Design Implications
LEAF LENGTH	2L = 1320.8 mm
NUMBER OF LEAVES (n)	N = 7
LEAF THICK NESS (t)	T = 9.525 mm
LEAF WIDTH (b)	B = 63.5 mm

Table 6: Considered variables of leaf spring.

Material Steel	State	Typical Physical Properties	
		ULTIMATE TENSIAL STRENGTH (Kg/Mm²)	TENSIAL YIELD STENGTH(Kg/Mm²)
50Cr 1	HARD	168 to 200	154 TO 175
50Cr 1 V 23	HARD	190 to 220	168to 189
55 Si 2 Mn90	HARD	182 to 206	168 to 192

Table 7: Physical properties of leaf spring materials.

Validation of the data

Stage 1: - Since the maximum load of 2W=6000N and Length of leaf 2L = 1320.8 mm where the effective length = 660.4 mm, and the width of leaf b = 63.5 mm, and stress taken = 377.5 N/mm^2, and number of leaves (Ng = 5, N_f=2) N=7 from where the appropriate thickness and the deflection that the springs can withstand are to be found, thus from the standard mathematical formulation;

Stress (377.5)=Equalized stress in the leaves = 18P*L / (b*t^2 (3nf+2ng))

Thus,

t^2=87.348

t=9.346 mm

Spring deflection = 12Pl3 /(b*t3 *E (3nf+2ng))

δ = 56.8223 mm

Standard sizes for automobile suspension springs;

(i) Standard nominal thickness are 3.2, 4.5, 5, 6, 6.5, 7, 7.5, 8, 9, 9.5, 10, 11, 12, 14 and 16 mm.

(ii) Standard nominal widths are 32, 40, 45, 50, 55, 60, 65, 70, 75, 80, 90, 100 and 125 mm (Table 8).

Thus the thickness being selected is 9.525 mm.

The standard data of the rear suspension of the DODGE car maker is taken as model to analysis (Table 9).

Thus now consider the following data:

Thickness (t) = 9.525 mm

Width (b) = 63.5 mm

Length (effective length) = 660.4 mm

Then the design data is validated by comparing the stress values and the deflection values within the limits are not. And found that the stress, similar to above from the above-mentioned mathematical calculations.

Stress ( ) = Equalized stress in the leaves

= 18P*L / (b*t2 (3nf+2ng))

= 18*3000*660.4 / (63.5*9.5252 (3*2+2*5))

σ = 363.447 N/mm^2

Spring deflection = 12Pl3 / (b*t^3 *E (3nf+2ng))

δ = 51.955 mm

Working stress	377.5 N/mm^2
Young modules	2.1E5 N/mm^2
Density	7.86E-6 kg/mm^2
Poison ration	0.3

Table 8: Standard sizes of leaf springs.

Maker	Location of the spring	Width of leaves(b)	Thickness of the leaves (t)	Length of main leaf	Total no of leaves in the assembly
DODGE	Rear	2.5 in (63.5 mm)	3/4 in (9.5 mm)	52 in (1320.5 mm)	7(n_g-5,n_f =2)

Table 9: Standard data for considered model.

Discussion: Thus the data for the parametric study of (DODGE car manufactures) with the data of Thickness (t) = 9.525 mm, Width (b) = 63.5 mm, Length (effective length) = 660.4 mm are achieved that the values are within the limits of the design. Now considering this data the model have to check by using fem nonlinear analysis package to achieve the results.

Static analysis of steel leaf spring

Elements taken for modeling leaf spring: Since the leaf spring is of symmetric section only half section is taken into consideration. The symmetric model leaf spring is modeled in ANSYS by using Cartesian coordinates system. Since leaf spring was modeled as a solid, for 50cr1 steel material leaf spring modeling solid element named SOLID45is taken. It is the element which is having a higher order 3-D, 8-node element.

The element is defined by 8 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials.

The geometry, node locations, and the coordinate system for this element are shown in the figure. In addition to the nodes, the element input data includes the orthotropic or anisotropic material properties. Orthotropic and anisotropic material directions correspond to the element coordinate directions (Figure 11).

Element taken for modeling the composite leaf spring: Element taken for modeling the composite leaf spring is SOLID191. It is a layered solid element of the 20-node structural solid designed to model layered thick shells or solids. The element allows up to 100 different material layers. If more than 100 layers are required, the elements may be stacked. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. SOLID191 has stress stiffening capabilities. The element is defined by 20 nodes, layer thicknesses, layer material direction angles, and orthotropic material properties. For any material, shear modules G_{XZ} and G_{YZ} must be within a factor of 10,000 of each other.

To reorient the elements (after automatic meshing) you should use EORIENT. With EORIENT, you can make SOLID191 elements match an element whose orientation is as desired, or set the orientation to be as parallel as possible to a defined axis. The element z-axis is defined to be normal to the reference plane, which may be curved. The default element x-axis is the average projection of side I-J and side M-N onto the reference plane. The total number of layers (up to 100) must be specified (NL). If the properties of the layers are symmetric about the

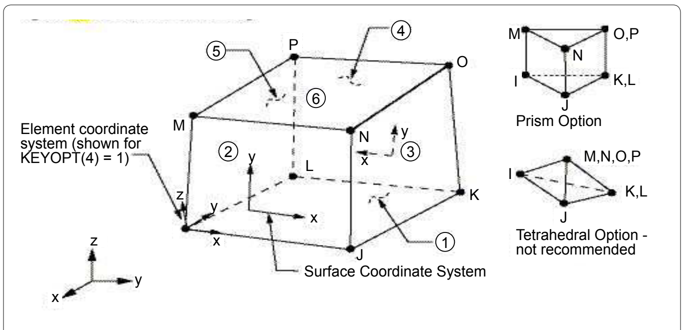

Figure 11: Solid 45 8-node geometry.

midthickness of the element (LSYM = 1), only half the properties, up to and including those of the middle layer (if any), need to be entered. Otherwise (LSYM = 0), the properties of all layers should be entered. The material properties of each layer may be orthotropic in the plane of the element. This layered191 is specified for the modeling of the composite leaf spring (Figure 12).

Performing the contact analysis: Contact problems are highly nonlinear and require significant computer resources to solve. It is important to understand the physics of the problem and take the time to set up the model to run as efficiently as possible. Contact problems present two significant difficulties.

- In generally the regions of contact between the bodies cannot be predicted until we run the problem. Depending on the loads, material, boundary conditions, and other factors, surfaces can come into and go out of contact with each other in a largely unpredictable and abrupt manner.
- Most contact problems need to account for friction. There are several friction laws and models to choose from, and all are nonlinear. Frictional response can be chaotic, making solution convergence difficult.

Basic step for analysis : The basic steps for performing a typical contact analysis are:-

- Create the model geometry and mesh
- Identify the contact pairs
- Designate contact and target surfaces
- Define the target surface
- Define the contact surface
- Set the element key options and real constants
- Define/control the motion of the target surface
- Apply necessary boundary conditions
- Define solution options and load steps
- Solve the contact problem
- Review the results

Geometric model of the leaf spring

The ultimate purpose of a finite element analysis is to re-create mathematically the behavior of an actual engineering system. In other words, the analysis must be an accurate mathematical model of a physical prototype. In the broadest sense, this model comprises all the nodes, elements, material properties, real constants, boundary conditions, and other features that are used to represent the physical system.

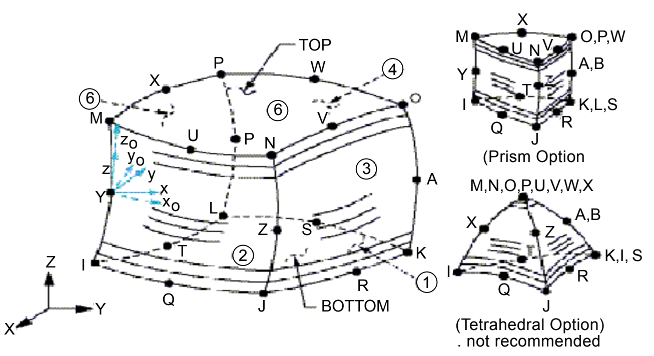

Figure 12: Layered191 20-node geometry.

Modeling procedure for steel leaf spring

- First create the key point 100 at the origin, i.e., x, y, z =(0,0,0).
- Create the another key point 200 at some arbitrary distance in Z- direction, say x, y, z=(0,0,200).
- Join the above two key points to 100 and 200 to get the reference axis.
- By using data from the mathematical analysis create a key point 1 with a distance of radius of curvature. R1 in vertically downward distance i.e., x, y, z = (0, -R1, 0).
- Note R1 = 4425.78, and R2 = R1+thickness (t=9.525).
- Similarly construct the remaining key points 2,3, 4 corresponding to R2, R3, R4 so on.
- Now join the key points sequentially as 1&2, 2&3. …till the last pair.
- Extrude the above lines with respect to the reference axis stated in step 3.
- Extrude line 1 with angle (Ø1=8.58) which give s area A1.
- Extrude line 2 with angle (Ø1=8.53), which gives area A2.
- Repeat the process till all the leaves are formed.
- And finally extrude the area to the width 63.5 mm along y-axis to form 3-d solid leaf springs. As shown in Figure 13.
- The obtained volumes are meshed for defining the nodal solutions.

To perform the contact analysis we first need to develop the appropriate geometric model that is capable of accomplishing the contact pairs between them. Thus SOLID45 is considered for our 3-D modeling of leaf spring structures, the geometric data for the modeling is obtained from the Table 2 and all other properties remains same to that of the above case, the next important case is the identification of the contact surfaces as source and target surface. So according to the defined conditions we had considered TARGE169, CONTA175 as the perfect contact pair for our analysis. Improper selection of the contact pairs may lead to solution convergence difficult thus the selection of the contact pair is shown in Figure 14 TARGE169. The element key options and real constants. The most important part in the contact analysis is the perfect selection of the element key options and real constants.

Determining contact stiffness

All contact problems require stiffness between the two contact surfaces. The amount of penetration between the two surfaces depends on this stiffness. Higher stiffness values decrease the amount

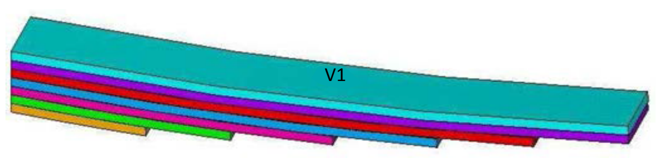

Figure 13: Leaf spring model in ANSYS.

of penetration but can lead to ill-conditioning of the global stiffness matrix and to convergence difficulties. Ideally, we want a high enough stiffness that contact penetration is acceptably small, but a low enough stiffness that the problem will be well-behaved in terms of convergence or matrix ill-conditioning. To arrive at a good stiffness value, we need to try the following procedure as a "trial run".

- Use a low value to start. In general, it's better to underestimate this value rather than overestimate it. Penetration problems resulting from a low stiffness are easier to fix than convergence difficulties that arise from a high stiffness.
- Run the analysis up to a fraction of the final load (just enough to get the contact fully established).
- Check the penetration and the number of equilibrium iterations used in each substep. If the global convergence difficulty is caused by too much penetration FKN may be underestimated or FTOLN may be too small. If the global convergence requires many equilibrium iterations for achieving convergence tolerances of residual forces and displacement increments rather than penetration, FKN may be overestimated.
- Adjust FKN or FTOLN as necessary, and run the full analysis.

Note: If the penetration control becomes dominant in the global equilibrium iterations (if more iterations are used to converge the problem to within the penetration tolerance than to converge the force residuals), you may increase FTOLN to permit more allowable penetration or increase FKN.

When the contact stiffness is too large (for example, 1016), the machine precision may not guarantee the good conditioning of the global stiffness matrix. You should scale the force unit in the model if possible.

The normal and tangential contact stiffness can be updated during the course of an analysis. Thus finally we end up with the following key options which well suits our analysis condition is tabulated in Figure 13.

Now the contact pairs are obtains one by one between all the layers are shown in Figure 15.

Now after formulating the appropriate contact pairs the nest stage is Apply necessary boundary conditions, and the loads which is shown (Figure 16).

Modeling procedure for composite leaf spring

Various kinds of composite leaf spring have been developed. In the case of multi-leaf composite leaf spring, the interleaf spring friction plays a spoil spot in damage tolerance. It has to be studied carefully. The following cross-sections of leaf spring for manufacturing easiness are considered.

- Constant thickness, constant width design
- Constant thickness, varying width design
- Varying thickness, constant width design

In the present work, only a leaf spring with varying thickness, constant width design is analyzed. The composite leaf spring is modeled by following steps:

- First the coordinate system is changed to global cylindrical coordinate system since the leafs are in a curved shape.
- By using the some geometric data key points are created in cylindrical coordinate system.
- For this a solid element LAYERED191 is taken which will form layers for placing the fibers.
- For giving the fiber direction a real constant set for LAYERED191 is assigned with 90^0 which will place the fibers along the leaf length direction.
- For rotating the fibers in the leaf arc direction a new coordinate system for fiber is created which includes a new local coordinate system for fibers.
- Each leaf is meshed with 10 layers which the fibers are kept in the leaf length direction.
- Such that each layer is arranged with thickness of 0.9525 mm.
- And remaining process like contact pair, constrains, loading are followed the same as steel leaf spring.

The analysis part is carried out on three stages which will give a clear idea about the suspension system of the automobile industry.

Stage 1: First to validate the design parameters the load applied on the leaf spring is varied. At different loads the response of the leaf spring is checked with the theoretical calculations.

Stage 2: Secondly by varying the normal penalty stiffness (FKN) value the response of the steel leaf spring is analyzed.

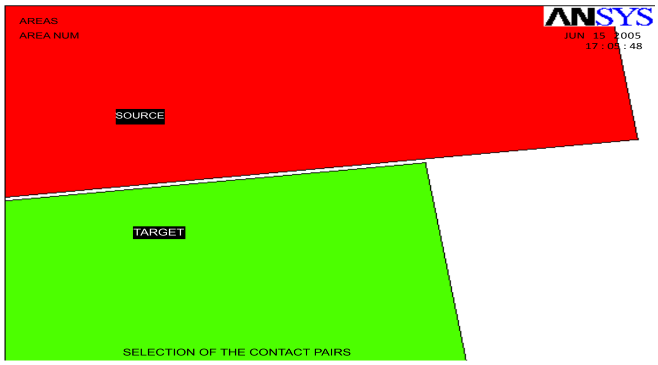

Figure 14: Selection of contact pairs (top source CONTA175 and bottom as target.

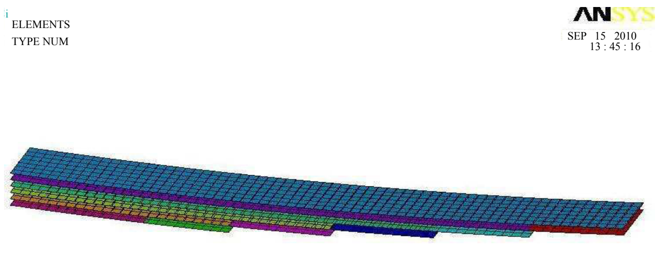

Figure 15: Contact pairs between different leafs.

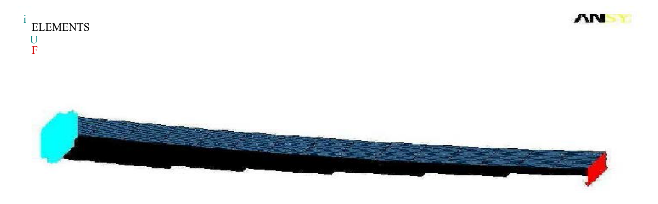

Figure 16: Boundary conditions and loading.

Stage 3: The same variation are carried for the composite leaf springs and a comparatively checked the stability for the stresses and deflections with respective to the steel design parameters.

Finally to replace the steel leaf spring with the composite material a model which will withstand the loads, stresses, and deflections produced at the actual working situation (Figures 17 and 18).

Model analysis of composite leaf spring

- In the preprocessor, Element type selected was SOLID45. Material properties selected in material model were structural - linear - elastic. Material models - structural - density, enter density.

In the mesh section the volume was meshed with SOLID45 3-d 8-node solid elements:

- In the solution section, the loads were defined as follows Constraints on displacements were given.
- Analysis type - new analysis, select modal analysis. Under analysis options, Block Lancozs and enter number of modes to extract as 5 and start frequency as 1 and end frequency as 10. Solution - solve - current LS.
- In general postprocessor, we go to read results and then we get different mode shapes by entering, first set Utility menu> plot controls> animate> mode shape.

Results and Discussions

The finite element model is generated in the ANSYS 12 software and the stresses, deflections, and the contact effect are obtained. The results are taken for both the conventional material (50cr1 steel) and composite material (E-Glass/Epoxy) of the leaf spring. The effect of the steel and E-Glass/Epoxy are taken into consideration. The static and model analyses are mainly considered for analysis the leaf spring. This will clearly explain the working efficiency of the leaf spring.

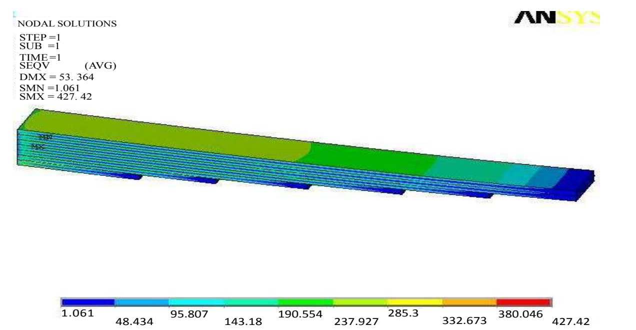

Figure 17: Static result for 50cr1 steel at 9.525 mm leaf thickness.

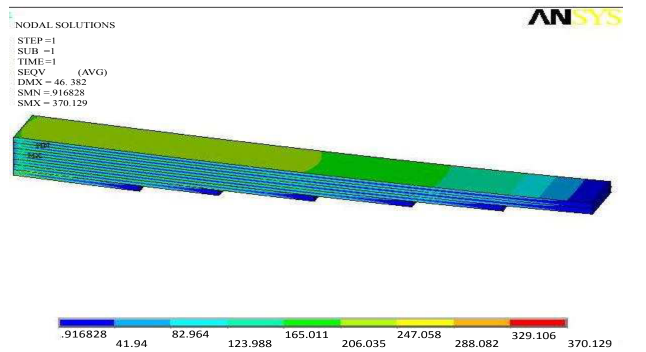

Figure 18: Static analysis of E- Glass/Epoxy at 8 mm leaf thickness.

In this work the analysis is carried out by three variations as follows.

Case 1: The variation of load applied on the leaf spring.

Case 2: The variation of normal penalty stiffness (FKN).

Case 3: The variation of the thickness of the leaf springs.

Case 4: performing the model analysis.

Case 1: The verition of the load

In this case by varying the load applied on the leaf spring there is more response in deflection, stresses, and contact effect. The following responses are plotted below (Figure 19).

In the figure 19, it is observed that when varying load the maximum deflection of the leaf spring for the steel material and the composite material are compared. Since the steel material having less young's modules than the E-Glass/Epoxy material it shows more deflection.

In the figure 20 the maximum stresses developed in the leaf spring at different loads are compared for the steel material and composite material. Here it is observed that the steel material is having higher stress than the composite material.

In the above graphs figure 21 it can clearly observe that the stiffness of the steel leaf spring and composite leaf spring are compared, the composite material is having more stiffness based on their material properties like young's module, poisons ratio etc.

In the figure 22 since the leaf springs are in surface to surface contact there will be a large scope of penetration between the leafs. The penetration between the leafs for the steel material and composite material is compared. The steel material is having more penetration this is based on the applied load and material propertied. This is mainly

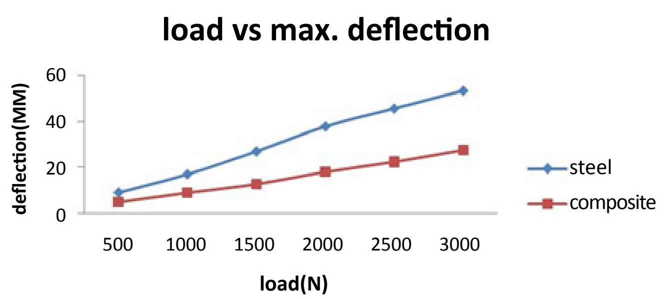

Figure 19: Variation of *δmax* with respective to *W* variation.

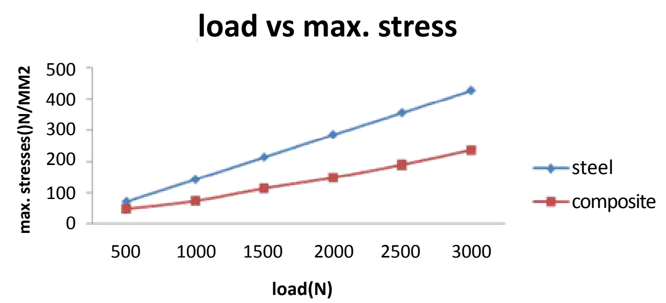

Figure 20: Variation of *σmax* with respective to *W* variation.

based of the contact type between the bodies.

In the Figure 23 the contact stress for the steel and composite material is plotted. It is observed that the stress produced in between the contact surface of the leafs are gradually increasing by varying the load applied on it. It states that the load is effecting the contact status between the surfaces.

From the Figure 24 it is observed that by varying the load applied on the leaf spring the contact sliding between the leafs is gradually increasing for both steel and composite material. If the sliding distance is more that means the load is effecting the body to deflect more. This will offer more stress in the body.

Case 2: The verition of the contact stiffness

The contact stiffness (or normal penalty stiffness) is the most important parameter affecting both accuracy and convergence behaviour of the contact nonlinear problems. for determine contact stiffness value for the problem more considerations like type of contact pair, friction coefficient, sliding distance etc. are should be specified carefully. The responses of the leaf spring like deflections, stresses, and contact parameters are plotted below (Figures 25 and 26).

In Figure 25 it is observed that the maximum deflection for the steel material is decreasing with a minute variation up to certain limit. This states that contact stiffness is effecting the max deflection of the leaf spring up to certain limit and getting constant.

The Figure 26 explains the same response for the maximum deflection for the composite material is decreasing with a little variation up to certain limit. This states that contact stiffness is effecting the max deflection of the leaf spring up to certain limit and getting constant (Figures 27 and 28).

From the Figure 27 it is observed that the maximum stress developed in the steel leaf spring gradually increased for certain limit but after certain increment of contact stiffness value the stress is being

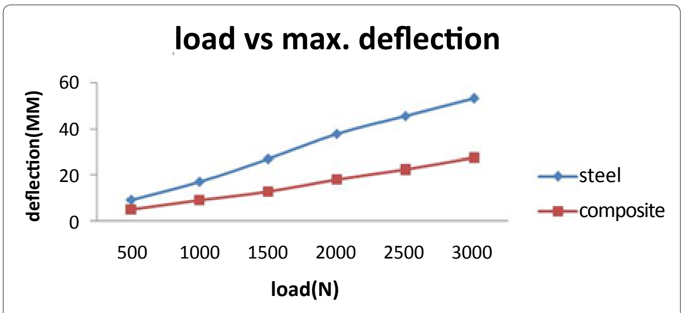

Figure 21: Variation of *K* with respective to *W* variation.

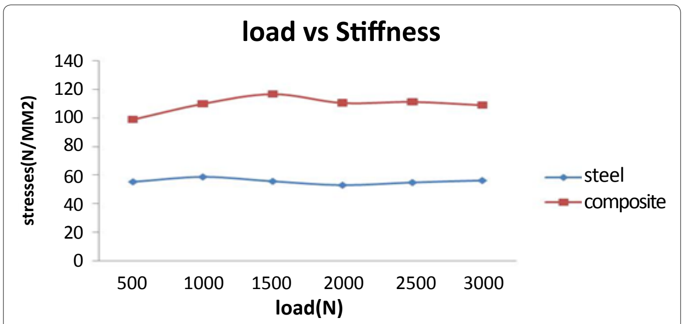

Figure 22: Variation of pene with respective to W variation.

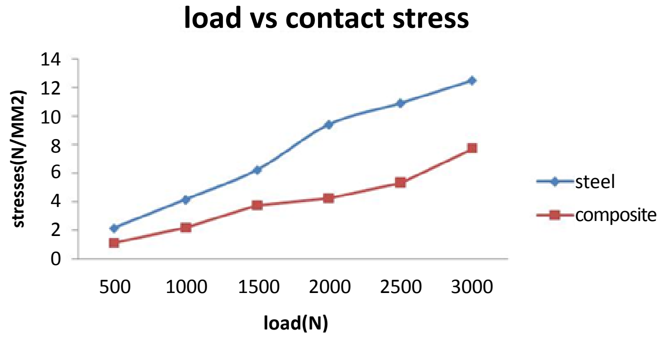

Figure 23: Variation of *stot* with respective to *W* variation.

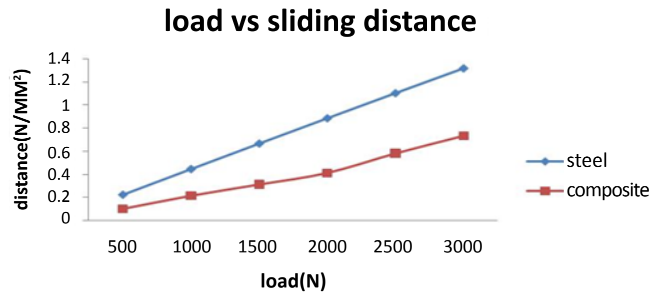

Figure 24: Variation of *slid* distance with respective to *W* variation.

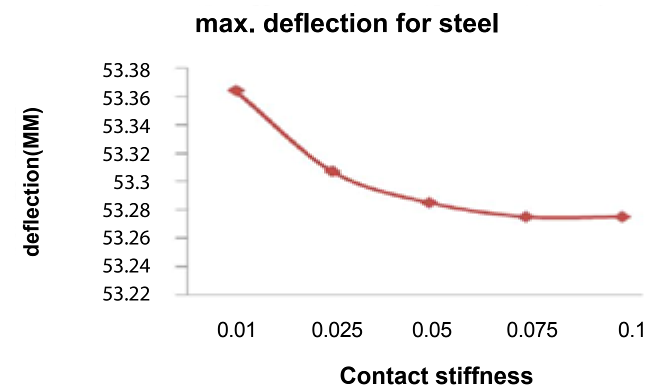

Figure 25: Variation of *δmax* with respective to *FKN* variation for steel.

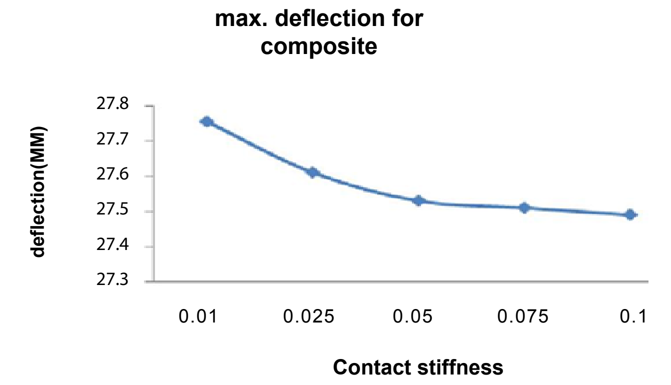

Figure 26: Variation of *δmax* with respective to *FKN* variation for composite.

constant. This state that the contact stiffness is not much affecting the leaf springs upto certain limit.

In this Figure 28 the some behavior is observed for the composite material of leaf spring here it is having a slit decrement is observed. Since it is having high body stiffness so it is being dominating the contact stiffness developed by the contact stiffness maintained between the leaf (Figures 29 and 30).

In the above graphs figure 29 it can clearly observe that the stiffness of the steel leaf spring by varying the contact stiffness value, the steel leaf is having a slit increment in its body stiffness upto certain limit.

From the above graph Figure 30, it can clearly observe that the stiffness of the composite material is also being gradually increasing up to certain limit and after certain value of the contact stiffness the stiffness of the body is being constant. This state that the FKN value will affect the total stiffness of the composite leaf spring (Figure 31).

In this graph (Figure 31) the variation for the penetration of the both steel and composite leaf springs are plotted. This graphs state that the penetration distance for steel leaf spring and composite leaf spring having a little variation. The steel material is having high penetration distance than composite material this is due to the material property. Since the composite material is having the high young's module (Figures 32 and 33).

In this Figure 32 the variation of contact stress of the steel leaf spring is plotted. The value of the penetration is gradually increased while changing the contact stress value. In the graph Figure 33 the same response is observed for the composite leaf spring. Here the contact

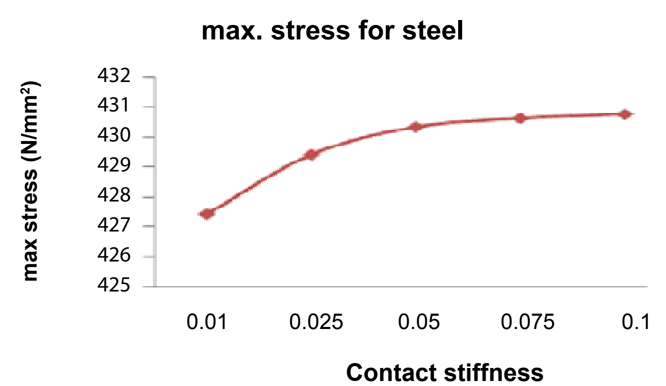

Figure 27: Variation of *σmax* with respective to *FKN* variation for steel.

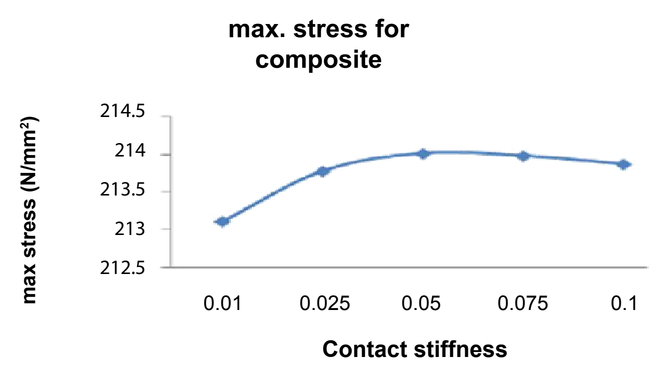

Figure 28: Variation of *σmax* with respective to *FKN* variation for composite.

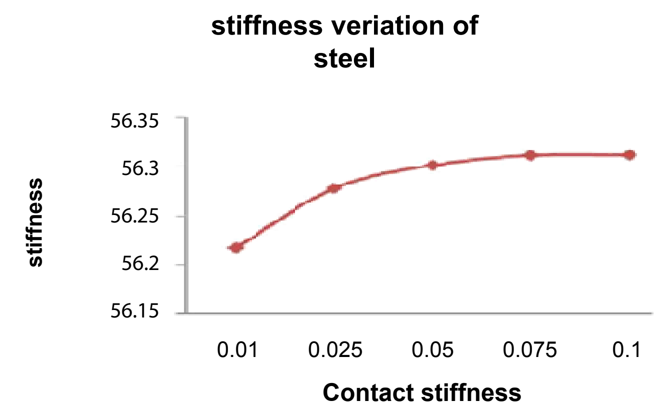

Figure 29: Variation of *k* with respective to *FKN* variation for steel.

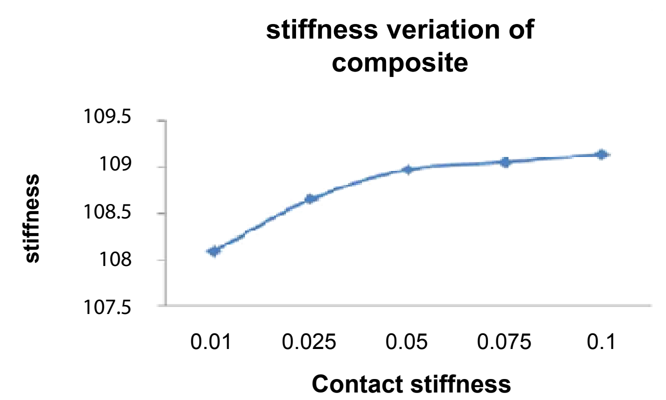

Figure 30: Variation of *k* with respective to *FKN* variation for composite.

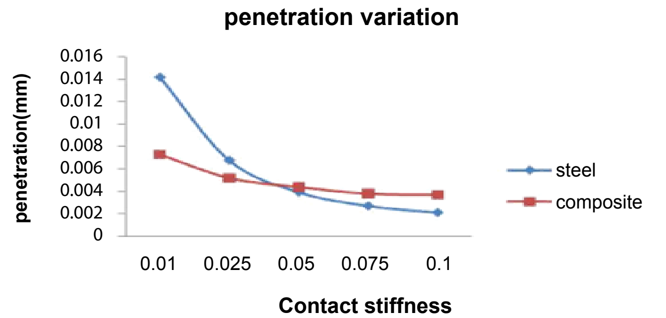

Figure 31: Variation of pene with respective to *FKN* variation for both steel and composite.

stress value is gradually increasing while changing the contact stiffness value (Figure 34).

In the figure 34 by varying the normal penalty stiffness value the contact sliding distance is having a large variation for both steel and composite leaf spring. From the graph the composite material is having less sliding distance rather than the steel material for the leaf springs. This state that the composite having a high resistance for deflection.

Case 3: The verition of the leaf thickness

In this case the main point is by varying the thickness of the composite leaf spring a correct replacement for the actual steel leaf

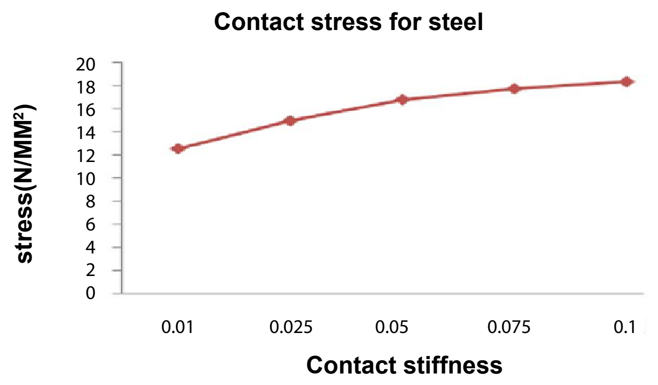

Figure 32: Variation of *stot* with respective to *FKN* variation for steel.

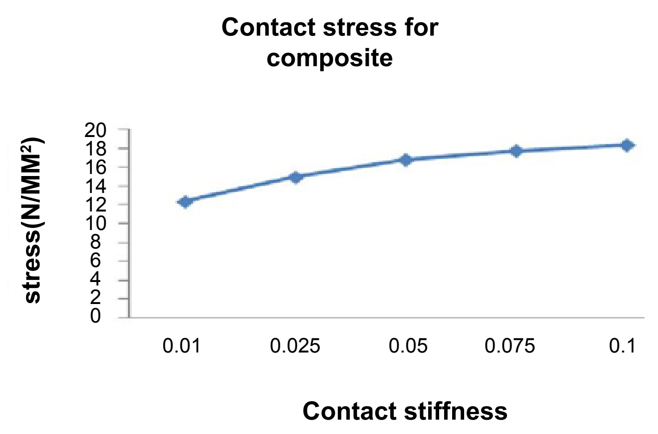

Figure 33: Variation of *stot* with respective to *FKN* variation for composite.

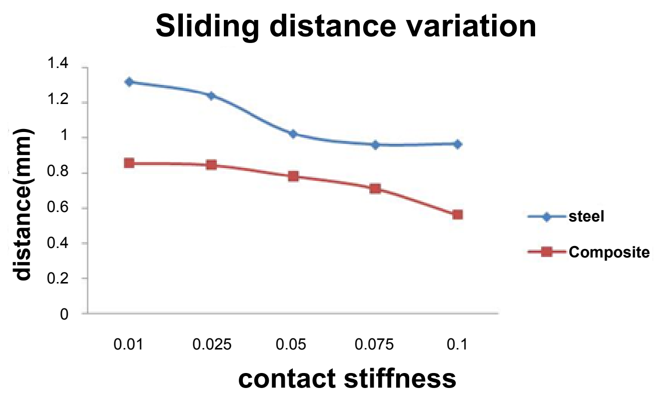

Figure 34: Variation of sliding distance with respective to *FKN* variation for both steel and composite.

spring is to be found. For that the variation of thickness for the composite leaf spring is carried out. Here the main consideration is to withstand the load applied on it, to maintain the considerable deflection, stresses stiffness and contact behaviour. These all parameters are compared with the standard 9.525 mm thick 50cr1 steel material leaf spring (Table 10).

From the above table by increasing the leaf thickness the maximum deflection has been gradually showing a decreasing manner in the similar way the stresses produced in the leaf are also having a decrement order. But our aim is to find the correct alternative composite leaf spring. On comparing the deflection, stresses and stiffness of the composite leaf spring at different thickness with the standard 50cr1 steel leaf spring the correct alternative is found. That is of 8 mm thick composite leaf spring. Here the composite leaf spring is having higher stiffnes than the steel leaf spring. The following table will clearly explain the difference of their deflection, stress and stiffness (Table 11).

Here from the table the deflection for composite leaf spring has be reduced to 13.850%, and the stresses along x-axis is 30.71%, along y-axis is 34.147%, along z-axis is 26.960% has been reduced. On considering the maximum stresses it is reduced upto 29.98% than the steel leaf spring. This state that 8 mm composite leaf spring is having more barring capacity than the 9.525 mm steel leaf spring.

Model analysis of leaf spring

The natural frequency of the 50cr1 steel material leaf spring and composite material leaf springs are carried out. The natural frequencies obtained for both leaf springs are noted below (Table 12).

From the above data it is absorbed that, since the composite material is having high stiffness it is exhibiting a high frequency. To provide ride comfort to passenger, leaf spring has to be designed in such a way that its natural frequency is maintained to avoid resonant condition with respect t road frequency. The road irregularities usually have the maximum frequency of 12Hz. Therefore the leaf should be designed to have a natural frequency, which is away from 12Hz to avoid the resonance. The first natural frequency of the composite material is of 18.547Hz. This is having 79% greater than the steel leaf spring frequency. And the first natural frequency of the composite leaf spring is nearly 1.7 times greater than the maximum road frequency and therefore resonance will not occur, and it provides improved ride comfort.

Comparison of leaf spring masses

The automotive industry is exploring composite material technology for structural components construction in order to obtain the reduction of weight without decrease in vehicle quality and reliability. Actually, there is almost a direct proportionality between the weight of the vehicle and its fuel consumption, particularly in city driving. From the above consideration of the composite leaf spring the weight was reduced. Here the 50cr1 steel leaf spring has 29.19 kg and the composite leaf spring has 8.11 kg. There for the weight is having 69.34% reduction has been achieved (Table 13).

Conclusion

Design and analysis of 50cr1 steel leaf springs and E-Glass/Epoxy composite leaf spring has been carried out. By introducing the contact pair in between the leafs a non-linear static analysis is done. In this it was found that the composite material leaf spring having a lesser stress of 52.65% than the steel leaf spring, and having higher stiffness of 49.943% than steel leaf spring. Since the composite material is having the higher stiffness than required for the smooth ride, it is re-designed by reducing the thickness of the leaf. On this point the composite material leaf spring thickness is reduced from 9.525 mm to 8 mm thick. This shows a stress difference of 29.981% and stiffness 12.951% at the standard deflection.

Secondly both the steel and composite material leaf springs are analyzed by varying the contact stiffness. In this the stresses, deflections and stiffness of both steel and composite leaf springs are having a variation up to certain limit. This from the results it is found that the contact stiffness value is 0.1 is being perfect value to analyses the leaf

Leaf Thickness	Max. deflection	σ_x	σ_y	σ_z	Von. mises stresses	Stiffness
6	98.238	654.52	73.78	190.23	639.75	30.532
6.5	81.145	465.9	75.28	168.32	462.82	36.97
7	66.09	399.81	68.64	111.64	380.86	45.392
7.5	58.406	375.01	51.93	103.97	357.49	51.36
8	46.382	316.45	316.45	87.801	370.129	64.69
8.5	37.741	266.79	40.37	85.98	264.174	73.22
9	31.698	248.76	38.98	72.45	247	94.65
9.5	27.215	230.9	35.065	66.91	217.98	110.23

Table 10: Variation of thickness for composite material.

	STEEL	COMPOSITE	% REDUCED
leaf thickness	9.525	8	16.01
Max. deflection	53.839	46.382	13.85
σ_x	452.76	316.45	30.71
σ_y	68.754	316.45	34.147
σ_z	120.21	87.801	26.96
Von mises stresses	430.78	370.129	29.98
Stiffness	56.3115	64.69	12.951

Table 11: Comparison of steel and composite leaf spring.

Mode	9.525 mm thick Steelleaf spring frequencies (Hz)	8 mm thick Composite leaf spring frequencies(Hz)
1	3.7999	18.547
2	3.9924	18.791
3	4.0868	21.543
4	5.3362	27.423
5	8.0172	32.091

Table 12: Mode frequencies of leaf spring.

Material	Density(Kg/mm^3)	Mass(Kg)
Steel	7.80E-06	29.191
Composite	2.60E-06	8.654
	% of weight reduced	69.48

Table 13: Mass of leaf spring.

spring which are in contact pair. In this the contact stresses, sliding distances, and penetration values are main points which should be considered for analyzing the leaf springs.

On reducing the thickness of the leaf spring it is found that the weight is reduced. The conventional multi-leaf spring weights about 29.191 kg but whereas the E-Glass/Epoxy multi-leaf spring weighs only 8.654 kg thereby weight reduction of 69.48% has been achieved.

Thirdly a simulated model analysis is carried out and found that the since the 8 mm thickness leaf spring is having a first natural frequency of 18.547 Hz which is lesser than the road irregularity maximum frequency(12Hz) the consideration taken is in safe condition.

Acknowledgement

I express my deep sense of gratitude and regards to a Mr. SD. Abdul Kalam, Assistant professor, Department of Mechanical Engineering, P.V.P Siddhartha Institute of Technology, Vijayawada for their keen interest, constant encouragement and invaluable guidance during the course of the work.

I express my deep sense of gratitude to our beloved Principal Dr. K. Srinivasu, and Dr. K. Sivaji Babu, Head of the Department for providing the necessary facilities to carry out the present research work.

I express my hearty and sincere gratefulness to Dr. V. Bala Krishna Murthy, mechanical department for giving a valuable suggestion for extending my project work.

I express our thanks to our programmer Mr. Rajendra Prasad in CAD LAB, Department of Mechanical Engineering for providing lab during the entire course of our work.

Finally I thank others who aided me directly or indirectly for successfully completing this project.

References

1. Kumar MS, Sabapathy V (2007) Analytical and experimental studies on fatigue life prediction of steel and composite multi-leaf spring for light passenger vehicles using life data analysis. Materials Science 13: 2.
2. https://hps.hs-regensburg.de/~rig39165/paper/Leaf_Spring.pdf
3. http://www.simulia.com/forms/world/pdf2002/qin.pdf
4. Manjunath HN, Manjunath K, T Rangaswamy (2007) Static analysis and fatigue life prediction of steel and composite leaf spring for light commercial vehicles. International Journal of Engineering Research 3: 422-425.
5. Mohammad Hosain, Moazzami Goudarzi (2006) The effects of replacement of steel leaf spring with a composite one on different aspects of the vehicle dynamics. International Conference on Advances in Mechanical Engineering.
6. Shankar GSS, Vijayarangan S (2006) Mono composite leaf spring for light weight vehicle – design, end joint analysis and testing. Materials Science 12: 3.
7. Pandey PC (2004) Composite materials.
8. Kurmi RS, Gupta JK (2005) Machine Design. Eurasia publishing house, Delhi, India.
9. Gallimore A, Muelles M (2002) Conversations with leaf spring manufacturers Guillermo Mezzatesta's. Design Journal, Mexico.
10. SAE (1982) Manual on design and applications of leaf springs – SAE HS J788 (4th edn) Society of Automotive Engineers. SAI Global Limited, Sydney NSW 2001, Australia.
11. http://www.irccyn.ec-nantes.fr/~martinet/Mobrob/FundamentalsofVehicleDynamics.pdf