Design and Finite Element Analysis of Leaf Spring Using Different Material Properties

Sagar Manchanda, Beant Singh and Gurmeet Singh
Dept. of Mechanical Engineering, P.C.E.T Lalru, Punjab, India
sagarmanchanda15@gmail.com*; +91 8950546976

Abstract

Automotive leaf spring is the main load carrier and energy absorbing component in a vehicle. In today’s modern world, fuel consumption is considered as a serious issue. Keeping all these constrains in consideration, strong and light weight material should be used for leaf spring design. Composite material would help reducing weight and improve fuel consumption without sacrificing the safety of the vehicle. Modification in the existing leaf spring design and material selection should be done at every stage of the analysis depending upon the simulation results for the safe design. In the present study, leaf spring has been designed using computer aided design (CAD) programme and analyzed using a finite element analysis (FEA) programme ANSYS, with different material (steel and composite) properties. Results of shear stress, shear strain, deformation and weight obtained from ANSYS has been taken into consideration in this study for final conclusion.

Keywords: Automotive leaf spring, fuel consumption, composite material, CAD programme, ANSYS.

Introduction

The suspension leaf spring is one of the potential items of automobile vehicle as it helps to improve riding quality of automobile vehicle. The main purpose of springs is to absorb, store and release energy. Leaf plays a vital role in supporting lateral loads, shock loads, brake torque, and driving torque. Advantages of leaf spring over helical spring are that the ends of the springs are guided along a definite path so as to act as a structural member in addition to shock absorbing device. This is the reason why leaf springs are still used widely in a variety of automobiles. Leaf spring made of composite materials made it possible to reduce the weight without any increase in load carrying capacity and stiffness of the leaf spring. Therefore, analysis of composite material leaf springs has become essential in showing the comparative results with conventional leaf springs. Yinhuans (2011) analyzed the mechanics of a composite leaf spring made from glass fiber reinforced plastics using the ANSYS software. Considering interleaf contact, the stress distribution and deformation were obtained. Taking the single spring as an example, comparison between the performance of the GFRP and the steel spring was presented. The comparison results showed that the composite spring had lower stress and much lower weight. Then, the automotive dead weight is reduced observably. Gebremeskel (2012) designed a single E-glass/Epoxy leaf spring and simulated following the design rules of the composite materials. And it was shown that the resulting design and simulation stresses are much below the strength properties of the material satisfying the maximum stress failure criterion.

This particular design was made specifically for light weight three wheeler vehicles. Its prototype was also produced using hand lay-up method. Raghavedra et al. (2012) compared laminated composite leaf spring and steel leaf spring with respect to weight, stiffness and strength. By employing a composite leaf spring for the same load carrying capacity, there was a reduction in weight of 73-80%, natural frequency of composite leaf springs are 27-67% higher than steel leaf spring and 23-65% stiffer than the steel spring. Anu Radha et al. (2013) attempted to study the material optimization for leaf spring. Hence, existing steel, carbon fiber and boron fibers were evaluated and optimized for the best one in concern with their performance. Boron fiber composite material was replaced with existing steel for master leaf spring. The master leaf spring was modeled in Pro/E (Wild Fire) 5.0 and analysis was carried out by using ANSYS 13.0 for better understanding. The objective of this project was to present modeling, stress analysis and material optimization of master leaf spring and comparison of deformation and stress results between steel leaf spring and composite leaf springs under same conditions. Yede and Sheikh (2014) presented a work of existing mono steel leaf spring of a Tata Bus taken for modeling and analysis. Finite element method had been implemented to modify the existing leaf spring with considering the dynamic loading. This work involved design and analysis of a conventional leaf spring under static and dynamic loading conditions. The 3D model was prepared in Creo 1.0 and then analysis was performed in the ANSYS 11.0 by considering same load in static and dynamic loading.
Chopade et al. (2015) considered the decreasing weight of light vehicle. The prominence of the experimentation was to reduce the overall weight of suspension system and improve load carrying capacity of the leaf spring by using the composite material. The design considerations for this study were stress and deflection and focus was also given on the application of FEA concept to compare two materials for leaf spring and propose the one having higher strength to weight ratio. Considering the above facts, in the present study, leaf spring has been designed using computer aided design (CAD) programme and analyzed using a finite element analysis (FEA) programme ANSYS, with different material (steel and composite) properties.

Materials and methods

Materials: The materials used to make leaf spring must meet certain requirements of geometry, strength and other limitations. The main criteria we took into consideration when choosing the material for the leaf spring are safety, cost and durability (Table 1 and 2). In a situation where vehicle would go through bumpy roads or if a vehicle met with an accident, the material used has to be sturdy enough to protect the driver from fatal injuries. Weight is a crucial factor and must be considered. The proper balance of fulfilling the design requirements and minimizing the weight is crucial to a successful design and it was considered carefully and made.

Experimental design: Linear static analysis was carried out on leaf spring to test spring under different loading conditions and to find out the resulting stresses and deformation (Fig. 1). Knowing how the current design reacts to different loading conditions; it would allow designers to make changes prior to physical prototyping. Following were the assumptions for static impact simulation:

1. The material is considered as isotropic and homogeneous.
2. Joints are assumed to be perfect joints.
3. The impact barrier is not deformable.
4. No inter leaf friction is considered.
5. Only vehicle and payload were applied vertically.

The Langrangian's method was used for meshing. Automatic contact procedure in ANSYS 15.0 was assumed for the complex interaction between the chassis and wall.

Results and discussion

Figure 2a-m represents the results of shear stress, shear strain, deformation and weight obtained when different material properties have been assigned to spring. Table 3 shows the compiled analysis of shear stress, shear strain, deformation and weight obtained from ANSYS. A load of 5300 N has been taken into consideration while performing analysis.
Fig. 2a to m representing the results of shear stress, shear strain, deformation and weight obtained when different material properties have been assigned to spring.

- a. Meshed model of leaf spring
- b. Von Mises stress contour of SLS
- c. Von Mises strain contour of SLS
- d. Total deflection of SLS
- e. Weight of steel leaf spring
- f. Von Mises stress contour of E-glass/E-poxy LFS
- g. Von Mises strain contour of E-glass/E-poxy LFS
- h. Total deflection of E-glass/E-poxy LFS
- i. Weight of steel leaf spring
- j. Von Mises stress contour of carbon polyamide
- k. Von Mises strain contour of carbon polyamide
- l. Total deflection of carbon polyamide
- m. Weight of carbon polyamide
Conclusion
The design and static structural analysis has been done. According to the present design, if we use composite materials (E-glass/E-poxy, carbon polyamide), carbon polyamide has been found to be suitable for mass reduction of 6.1 kg (approx. 80% than steel) moreover, stress, strain and deformations observed are almost similar which shows that the vehicle is light weight and can generate more torque and power.

References

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Steel</th>
<th>E-glass/E-poxy</th>
<th>Carbon polyamide</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load Applied (N)</td>
<td>5300</td>
<td>5300</td>
<td>5300</td>
</tr>
<tr>
<td>Shear Stress (Pa)</td>
<td>3.2106e8</td>
<td>3.2361e8</td>
<td>2.6501e8</td>
</tr>
<tr>
<td>Shear Strain (m/m)</td>
<td>0.0017039</td>
<td>0.066383</td>
<td>0.0063075</td>
</tr>
<tr>
<td>Deformation (m)</td>
<td>0.0046512</td>
<td>0.18673</td>
<td>0.02828</td>
</tr>
<tr>
<td>Weight (Kg)</td>
<td>7.725</td>
<td>1.968</td>
<td>1.564</td>
</tr>
</tbody>
</table>