EYE DESIGN ANALYSIS OF SINGLE LEAF SPRING IN AUTOMOTIVE VEHICLES USING CAE TOOLS

*Vinkel Arora*, Gian Bhushan and M.L. Aggarwal

1) Department of Mechanical Engineering ITM University, Sector-23A, Guragaon-122017, Haryana, India
2) Department of Mechanical Engineering National Institute of Technology, Kurukshetra, Haryana, India
3) Department of Mechanical Engineering YMCA University of Science &Technology, Faridabad, Haryana, India

*Author for Correspondence

ABSTRACT
This paper is focused on determination of better eye end design of single leaf spring used in light motor vehicle. The procedure of this work is to carry out computer aided design and analysis of conventional single leaf spring with actual design considerations and loading conditions. This conventional 65si7 spring steel leaf spring model with standard eye end and casted eye end are considered. The CAD model of the leaf springs is prepared in CATIA and analyzed using ANSYS. The standard eye and casted eye leaf springs are subjected to similar loading conditions. The CAE analysis of the leaf spring is performed for various parameters like deflection, von-mises stress, normal stress etc. The main objective of this work is to determine the better eye end design and reduce the time and cost related to actual experimental testing by providing a CAE solution.

Key Words: Computer Aided Engineering (CAE); Leaf Spring; 65Si7 Spring Steel; Static Loading

INTRODUCTION
In this work single leaf spring is modeled using dedicated modeling software CATIA and considering various eye design the stresses induced in the leaf spring are computed. As eye end plays a vital role during application of leaf spring like, eyes have the critical areas where the most stresses induced in a leaf spring. Thus by changing the design of eye, stresses can be reduced. For this purpose two different types of eye design for leaf spring analysis were considered. These two eye design are 1) Standard eye and 2) Casted eye.
Mouleeswaran et al [2007] describes static and fatigue analysis of steel leaf springs and composite multi leaf spring made up of glass fibre reinforced polymer using life data analysis. The dimensions of existing conventional steel leaf springs of a light commercial vehicle are taken and are verified by design calculations. Static analysis of 2-D model of conventional leaf spring is also performed using ANSYS 7.1 and compared with experimental results. Hawang W et al [1986] Fatigue of Composites – Fatigue Modulus Concept and Life Prediction Journal of Composite Materials. H. A. Al-Qureshi [2001] has described a single leaf, variable thickness spring of glassfiber reinforced plastic (GFRP) with similar mechanical and geometrical properties to the multileaf steel spring, was designed, fabricated and tested. J.J.Fuentes et al [2008] in this work, the origin of premature failure analysis procedures, including examining the leaf spring history, visual inspection of fractured specimens, characterization of various properties and simulation tests on real components, were used. Rajendran I. S. Vijayarangan [2002] A formulation and solution technique using genetic algorithms (GA) for design optimization of composite leaf springs is presented here. Gulur Siddaramanna et al [2006] explain the automobile industry has shown increased interest in the replacement of steel spring with fiberglass composite leaf spring due to high strength to weight ratio. Leaf springs industries working with 65Si7 spring steel are using a very low factor of safety for weight reduction. To achieve this, experimental testing is done to predict the spring rate, bending stress and deflection. Aggarwal M.L et al [2006] evaluated the axial fatigue strength of EN45A spring steel specimen experimentally as a function of shot peening in the conditions used for full-scale leaf springs testing in industries. S/N curves of the specimens are correlated with leaf springs curve in vehicles. The process is time consuming and costly. In the present work, a CAE system predicts various variables in complex assemblies of leaf springs and the results are compared with experimental testing. J.P. Hou et al [2007] explained the design evolution process of a composite leaf spring for freight rail application.

MATERIALS
The basic requirements of leaf springs steel is that the selected grade of steel must have sufficient harden ability for the size involved to ensure a full martensitic structure throughout the entire leaf section. In general terms higher alloy content is mandatory to ensure adequate harden ability when the thick leaf sections are used. The material used is 65Si7. The chemical composition of the material is shown below in Table 1.

<table>
<thead>
<tr>
<th>GRADE</th>
<th>C%</th>
<th>Si%</th>
<th>Mn %</th>
<th>P%</th>
<th>S%</th>
</tr>
</thead>
<tbody>
<tr>
<td>65Si7</td>
<td>0.61</td>
<td>0.79</td>
<td>0.014</td>
<td>0.035</td>
<td>0.024</td>
</tr>
</tbody>
</table>
1.1 Design Parameters
The design parameters are shown below in Table 2.

<table>
<thead>
<tr>
<th>Table 2: Design parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
</tr>
<tr>
<td>Material selected- steel</td>
</tr>
<tr>
<td>Young’s Modulus, E</td>
</tr>
<tr>
<td>HRC</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
</tr>
<tr>
<td>Tensile strength Ultimate</td>
</tr>
<tr>
<td>Tensile strength Yield</td>
</tr>
<tr>
<td>Leaf span</td>
</tr>
<tr>
<td>Spring stiffness</td>
</tr>
<tr>
<td>Density</td>
</tr>
</tbody>
</table>

2. Modeling and Analysis
The 2D drawing of leaf springs is shown in the Figure 4 below.

3.1 CAD Modeling
CAD modeling of any project is one of the most time consuming process. One cannot shoot directly from the form sketches to finite element model. CAD modeling is the base of any project. The finite element software will consider shapes, whatever is made in CAD model. Although most of the CAD modeling software have capabilities of analysis to some extent and most of finite element software have capabilities of generating a CAD model directly for the purpose of analysis, but their off domain capabilities are not
sufficient for large and complicated models which include many typical shapes of the product. The CAD models of the eye design were prepared in CATIA and the analysis and comparison of results are performed using Ansys. The CAD model of the standard eye and casted eye are shown in the Figure below.

3.2 Analysis Using Ansys
The CAD model of leaf springs is now imported into Ansys-11 as shown in figures below. All the boundary conditions and material properties are specified as per the standards used in the practical application. The material used for the leaf spring for analysis is structural steel, which has approximately similar isotropic behavior and properties as compared to 65Si7 spring steel leaf springs.
The procedure for performing analysis in ANSYS involves:

### 3.2.1 Specifying Joints
A joint is an idealized kinematics linkage that controls the relative movement between two bodies. Joint types are characterized by their rotational and translational degrees of freedom as being fixed or free. In this assembly two revolute joints are used between eye and pin. The joint rotation is 18° corresponding to no load camber angle of 162°.

### 3.2.2 Meshing
Meshing is the process in which your geometry is spatially discretized into elements and nodes. This mesh along with material properties is used to mathematically represent the stiffness and mass distribution of your structure. The default element size is determined based on a number of factors including the overall model size, the proximity of other topologies, body curvature, and the complexity of the feature. If necessary, the fineness of the mesh is adjusted up to four times (eight times for an assembly) to achieve a successful mesh. In this assembly SOLID92 mesh element is used for the results.
3.2.3. Setting Analysis Environment
A static structural 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. Static structure analysis takes into consideration some parameters, like material properties, loading conditions, support conditions, joints and contacts which are to be specified as the input to the pre processing of the analysis.

3.2.5. Setting Boundary Conditions
The boundary conditions are applied by taking into consideration the experimental loading conditions. The static loading condition of single leaf spring involves the fixation of one of the revolute joint and applying displacement support at the other end of leaf springs. Loading conditions involves applying a load at the centre of the main leaf. As per specifications the springs is drawn at flat condition, therefore the load is applied in downward direction to achieve initial no load condition. As no load assembly camber is 162°.

Figure 9: Meshed model of standard eye leaf
Figure 10: Meshed model of casted eye leaf

Figure 11: Static loading condition
4.0 Solution

4.1 Results for standard eye

Figure 12: Boundary condition in ANSYS

Figure 13: Total deformation

Figure 14: Normal stress

Figure 15: Von–Mises stress

Figure 16: Factor of safety
4.2 Results for casted eye

![Figure 17: Total deformation](image)

![Figure 18: Normal stress](image)

![Figure 19: Von –Mises stress](image)

![Figure 20: Factor of safety](image)

RESULTS AND DISCUSSION

Table 3: Result Comparison

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Standard Eye</th>
<th>Casted Eye</th>
<th>Variation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load</td>
<td>50 N</td>
<td>50 N</td>
<td>Nil</td>
</tr>
<tr>
<td>Deflection</td>
<td>109.62 mm</td>
<td>115.61 mm</td>
<td>5.4%</td>
</tr>
<tr>
<td>Von Mises Stress</td>
<td>773.34MPa</td>
<td>750.08MPa</td>
<td>3.0%</td>
</tr>
<tr>
<td>Normal Stress</td>
<td>654.12MPa</td>
<td>783.51MPa</td>
<td>19.08%</td>
</tr>
<tr>
<td>Factor of Safety</td>
<td>Maximum-15</td>
<td>Maximum-15</td>
<td>Nil</td>
</tr>
<tr>
<td></td>
<td>Minimum-0.381</td>
<td>Minimum-0.331</td>
<td>13.1%</td>
</tr>
</tbody>
</table>

a. From the above table it is observed that for same static load application, when the standard eye is replaced with casted eye, equivalent stress reduction of 2.9% and increase in deflection of 5.44% is achieved.

b. From the above table it is observed that the minimum value of factor of safety is reduced by 13.1% for casted eye.

c. It is also observed that there is increase in Normal stress by 19.08% in the case of casted eye.
Conclusion
This work involves design and analysis of single leaf spring under static loading conditions. The 3D model is prepared in CATIA and then CAE analysis is performed using ANSYS-11. From the results obtained from ANSYS, discussions have been made and it will be concluded that:
1. When same load is applied to standard and casted leaf spring equivalent stress reduction of 2.9% and increase in deflection of 5.44% is achieved. As maximum stress induced is below the yield stress therefore it is concluded that casted eye is also safe under given loading conditions.
2. At the same time bending stress for same load is increased by 19.08% in case of casted eye analysis as compared with standard eye. This may be observed because the actual material is 65Si7 but for CAE analysis structural steel is used.
3. As the minimum factor of safety is reduced by 13.1% in case of casted eye it is concluded that the area of minimum factor of safety will fail earlier in case of casted eye. Hence casted eye is not recommended.
4. It is concluded that CAE tools provides a cost effective and less time consuming solution in comparison with the experimental testing but the results may vary in the specified range.

ACKNOWLEDGEMENT
At the very last, I would like to acknowledge Friends Auto industries, Faridabad and their leaf spring testing division for their constant support and guidance.

REFERENCES
Research Article


Peiyong Qin, Glenn Dentel and Mikhail Mesh (2002). Multi-Leaf Spring and Hotchkiss Suspension. *ABAQUS Users’ Conference.*