A Comparative Study of CAE and Experimental Results of Leaf Springs in Automotive Vehicles

VINKEL ARORA¹
Department of Mechanical Engineering
ITM University, Sector-23A, Guragaon-122017, Haryana, India
vinkelarora@gmail.com

Dr. M.L AGGARWAL²
Professor & Dean, Department of Mechanical Engineering
YMCA University of Science & Technology, Faridabad, Haryana, India

Dr. GIAN BHUSHAN³
Associate Professor, Department of Mechanical Engineering
National Institute of Technology, Kurukshetra, Haryana, India

Abstract:
The work is carried out on the front end leaf spring of a commercial vehicle. The objective of this work is to carry out computer aided design and analysis of a conventional leaf spring, with experimental design considerations and loading conditions. This conventional leaf spring model consists of 37 parts. The material of the leaf spring is 65Si7. The CAD model of the leaf spring is prepared in CATIA and analyzed using ANSYS. The CAE analysis of the leaf spring is performed for the deflection and stresses under defined loading conditions, using ANSYS. The experimental and CAE results are compared for validation. Using CAE tools the ideal type of contact and meshing element is determined in leaf spring model.

Keywords: Computer Aided Engineering (CAE); Leaf Spring; 65Si7; Static loading.

1. Introduction

CAE tools are widely used in the automotive industries. In fact, their use has enabled the automakers to reduce product development cost and time while improving the safety, comfort, and durability of the vehicles they produce. The predictive capability of CAE tools has progressed to the point where much of the design verification is now done using computer simulation rather than physical prototype testing. CAE dependability is based upon all proper assumptions as inputs and must identify critical inputs. Even though there have been many advances in CAE, and it is widely used in the engineering field, physical testing is still used as a final confirmation for subsystems due to the fact that CAE cannot predict all variables in complex assemblies, therefore the validation of CAE results is important.

Mouleswaran et al [1] 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. The dimensions of an existing conventional steel leaf spring 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 [2] Fatigue of Composites – Fatigue Modulus Concept and Life Prediction Journal of Composite Materials. H. A. Al-Qureshi [3] has described a single leaf, variable thickness spring of glass fiber reinforced plastic (GFRP) with similar mechanical and geometrical properties to the multileaf steel spring, was designed, fabricated and tested. J.J.Fuentes et al [4] 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 [5] A formulation and solution technique using genetic algorithms (GA) for design optimization of composite leaf springs is presented here. Gulur Siddaramanna et al [6] 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. Therefore; the aim of this paper is to present a low cost fabrication of complete mono composite leaf spring and mono composite leaf spring with bonded end joints. J.P. Hou et al [7] explained the design evolution process of a composite leaf spring for freight rail application. Peiyong et al [8] describes that the Leaf spring design was mainly based on simplified equations and trial-and-error methods. The simplified equation models were limited to the three-link mechanism assumption and linear beam theory. This work
presents detailed finite element modeling and analysis of a two-stage multi-leaf spring, a leaf spring assembly, and a Hotchkiss suspension using ABAQUS. Muhammad Ashiqur et al [9] describes that the tapered cantilever beams, traditionally termed as leaf springs, undergo much larger deflections in comparison to a beam of constant cross-section that takes their study in the domain of geometric nonlinearity. This paper studies response of a leaf spring of parabolic shape, assumed to be made of highly elastic steel. 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 [10] 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 all variables in complex assemblies of leaf springs and the results are compared with experimental testing.

2. Experimental Setup

The leaf spring involves two full length leaves and seven graduated leaves, four packing which are made of 65Si7 material. This conventional leaf spring model consists of 37 parts which, includes two full length leave, seven graduated leaves. The remaining part involves four rebound clips of MS, four shim pipes of C.D.S.T/ERW, centre nut & bolt and bush of bronze. [10]The experimental setup consists of a full scale testing machine for leaf spring, jigs and fixture. The system consists of a hydraulic power pack to give a hydraulic pressure of 20.6 MPa with a flow rate of 210 lpm, which is sent to a hydraulic actuator to operate at a frequency of 0.3 Hz with the displacement specified by the alternating load. This involves applying the axial load on the leaf spring and measure the deflection and bending stress. Supavut, Chantranuwathana, et al [11] have simulated a leaf spring model. An experimental leaf spring model was verified by using a leaf spring test rig that can measure vertical static deflection of leaf spring under static loading condition.

![Figure 1 Full scale testing machine for leaf spring](image)

2.1 Material

The basic requirements of a leaf spring steel is that the selected grade of steel must have sufficient harden ability for the size involved to ensure a full martenstic 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 for the experimental work is 65Si7 . The chemical composition of the material is shown below in Table -1:-
### 2.2 Design Parameters

The design parameters are shown below in Table-2

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material selected- steel</td>
<td>65Si7</td>
</tr>
<tr>
<td>Young’s Modulus, E</td>
<td>$2.1 \times 10^5$ N/mm²</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.266</td>
</tr>
<tr>
<td>BHN</td>
<td>455-461</td>
</tr>
<tr>
<td>Tensile strength Ultimate</td>
<td>460 MPa</td>
</tr>
<tr>
<td>Tensile strength Yield</td>
<td>250 MPa</td>
</tr>
<tr>
<td>Leaf span</td>
<td>1450mm</td>
</tr>
<tr>
<td>Spring stiffness</td>
<td>220 N/mm</td>
</tr>
<tr>
<td>No Load Camber Angle</td>
<td>153°</td>
</tr>
<tr>
<td>Density</td>
<td>0.00000785 Kg/mm³</td>
</tr>
</tbody>
</table>

### 2.3 Loading conditions

The static Loading condition of the multi leaf spring involves the fixation of one of the revolute joint and applying displacement support at the other end of leaf spring. Loading conditions involves applying a load at the centre of the main leaf. As per specifications the spring 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 153°. The loading conditions are shown below in the Fig-2.
2.4 Experimental Load-Deflection Curve

A static load of 35KN (Full Load) and 17.5KN (Half load) is applied by a universal testing machine and the corresponding deflection and stress values are observed. A graph between load – deflection is plotted. The plot shown below in Fig 3 depicts a linear relationship between load and deflection for full as well as for half load.

![Experimental Load-Deflection curve](image)

**Figure 3 Load-deflection curve**

3.0 2D- drawing of Leaf spring.

For performing the CAE of leaf spring the 2D drawing of a leaf spring is converted into a part model using CATIA.

![2D drawing of leaf spring](image)

**Figure 4- 2D drawing of leaf spring**
3.1 CAD Modeling

CAD Modeling of any project is one of the most time-consuming processes. One cannot shoot directly from the form sketches to Finite Element Model. CAD Modeling is the base of any project. 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 model of the multi leaf spring structures also includes many complicated parts, which are difficult to make by any of other CAD modeling as well as Finite Element software. CAD modeling of the complete multi Leaf Spring structure is performed by using CATIA V5 R17 software. CAD model of leaf spring consist of total 37 different parts which are assembled together in assembly design to make a complete multi leaf spring model, out of all 37 parts, some parts are similar in shape & size. The CAD model of multi leaf spring used for analysis is shown in Fig-5 below:-

![Figure-5 CAD model of a leaf spring](image)

3.2 Analysis using ANSYS

The CAD model of leaf spring is now imported into ANSYS-11 as shown below in Fig-6. 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.

![Figure-6 CAD model imported in ANSYS](image)

The procedure for performing analysis in ANSYS involves:
3.2.1. Setting contact reign—Contact conditions are formed where bodies meet. When an assembly is imported from a CAD system, contact between various parts is automatically detected. In addition you can also set up contact regions manually. You can transfer structural loads and heat flows across the contact boundaries and connect the various bodies. Depending on the type of contact, the analysis can be linear or nonlinear. The differences in the contact settings determine how the contacting bodies can move relative to one another. This is the most common setting and has the most impact on what other settings are available. Most of these types only apply to contact regions made up of faces only. In this assembly the No separation contact is used for the analysis. It only applies to regions of faces. Separation of faces in contact is not allowed, but small amounts of frictionless sliding can occur along contact faces. In general CONTA 72 and TARGET 71 are used.

![Figure-7 No separation contact](image)

3.2.2. 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 27° corresponding to no load camber angle of 153°.

![Figure-8 Revolute joint between eye and pin](image)

3.2.3. 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 element is used for the results.

![Meshed model of leaf spring](image)

**Figure -9 Meshed model of leaf spring**

3.2.4. **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.
3.2.6 Solution

Figure 10 Boundary condition

Figure 11 Deflection at 17.5KN (half load)
Figure 12 Deflection at 35KN (full load)

Figure 13 CAE Load-deflection curve

4. Results and Discussions

CAE analysis of the leaf spring has been done and the results are compared with the experimental results as follows:-
4.1 For Static Load 35 KN

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Exp. Results</th>
<th>CAE Results</th>
<th>Variation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td>158 mm</td>
<td>156.15 mm</td>
<td>1.17%</td>
</tr>
<tr>
<td>Bending Stress</td>
<td>126 Kgf/mm²</td>
<td>141.56 Kgf/mm²</td>
<td>12.30%</td>
</tr>
<tr>
<td>Spring rate</td>
<td>221.5 N/mm</td>
<td>224.5 N/mm</td>
<td>1.35%</td>
</tr>
</tbody>
</table>

From the above Table -3 it has been observed that for the same static loading conditions, deflection in experimental & CAE results are 158mm and 156.15mm respectively. Bending stress for experimental results and CAE results is 126kgf/mm² and 141kgf/mm². the variation in deflection and bending stress is 1.17% and 12.30% respectively.

4.2 For Static Load 17.5 KN

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Exp. Results</th>
<th>CAE Results</th>
<th>Variation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection</td>
<td>79 mm</td>
<td>78.07 mm</td>
<td>1.1%</td>
</tr>
<tr>
<td>Spring Rate</td>
<td>221.5 N/mm</td>
<td>224.5 N/mm</td>
<td>1.35%</td>
</tr>
<tr>
<td>Bending Stress</td>
<td>48 Kgf/mm²</td>
<td>53.77 Kgf/mm²</td>
<td>12.02%</td>
</tr>
</tbody>
</table>

From the above shown Table-4, it has been observed that for the same static loading conditions, deflection in experimental results is 78 mm and deflection for CAE results is 78.07 mm. Bending stress for experimental results and CAE results are 48 kgf/mm² and 53.77kgf/mm². the variation in deflection and bending stress is 1.17% and 12.02% respectively.

5. Conclusion

This work involves design and analysis of a conventional 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, many discussions have been made and it will be concluded that:

1. When the leaf spring is fully/half loaded, a variation of 1.17% in deflection is observed among the Experimental & CAE value, which proves the validation of our CAD model and analysis.

2. At the same time bending stress for fully loaded, is increased by 12.30 % in CAE analysis as compared with experimental and for half loaded bending stress is increased by 12.02 %. This may be observed because the actual material is 65Si7 but for CAE analysis Structural steel is used.

3. The maximum equivalent stress is 172.5 MPa & 86.29 MPa for fully and half loaded leaf spring respectively, which is below the Yield Stress i.e. 250MPa. Therefore the design, is safe.

4. It is concluded that when CONTAT72, TARGET71 type of contact and SOLID 92 mesh element is used for CAE analysis the results are closer to the Experimental results. Therefore the CAD model can be used for fatigue loading under defined boundary conditions.

References


